WoodWorks®
Design Office

Sizer | Shearwalls | Connections | Database Editor

User Guide

For Canadian and U.S. version

This edition of the guide includes new features introduced in the U.S. Design Office 9

Canadian Wood Council
American Wood Council

Developed by
Acronym Software Inc.

ISBN 0-921628-52-8

1.0M0900, 1.0M1199, 0.5M1098, 0.5M198, 0.5M1206, POD1010

Limited Warranty

The American Wood Council and the Canadian Wood Council make no representations or warranties with respect to the contents of this user guide and specifically disclaim any implied warranties of merchantability or fitness for any particular purpose. The contents of this documentation may include technical inaccuracies or typographical errors.

The American Wood Council and the Canadian Wood Council disclaim all warranties with respect to the software contained on disk or in printed form, including all expressed or implied warranties of merchantability and fitness; and any stated or expressed warranties are in lieu of all obligations or liabilities of the American Wood Council and the Canadian Wood Council for damages, including, but not limited to special, indirect or consequential damages arising out of or in connection with the performance of the software.

As always, the engineer is ultimately responsible for his or her design.

Refer to Read Me for further information.

Version 2010
October 2010

Printed in Canada.

Table of contents

Welcome

What Is Design Office

WoodWorks® Design Office is an integrated design suite for engineering projects. The three programs that complement each other are Sizer, Shearwalls, and Connections.

This User Guide applies to both Canadian and USA versions of WoodWorks® software. Minor differences in the programs do exist and some screen captures may not do exactly as shown in this document.

This User Guide covers all three modules of the Design Office suite.

The purpose of this Guide is to help the beginner quickly and efficiently learn how to use all three modules of the software, and includes tutorials. More information is available in the help files that can be opened within the software. Refer also to www.woodworks-software.com.

Technical Support

If you have installation or performance problems, please contact WoodWorks® Support via one of the options listed below.

For questions about engineering assumptions, features and functions, please consult the online help which provides a keyword search feature.

The WoodWorks® website contains additional information which includes product news, frequently asked questions, maintenance releases, and updates for registered software owners.

Email: support@woodworks-software.com

Voice: 1-800-844-1275

URL: www.woodworks-software.com

Installing Design Office

System Requirements

The following are the requirements for running WoodWorks® Design Office on your computer:

- PC-compatible system with a minimum of:
- 1.0 GHz processor (recommended: 2.0 GHz or better)
- 512 MB or random access membory (RAM) (recommended 1 GB or more)
- 800 x 600/500 or higher video resolution (recommended: 1280 x 1024/768)
- Microsoft Windows 7, Vista, XP, and 2000 (32 and 64 bit)

Installation from CD

- Start up Windows
- Insert the WoodWorks® Design Office CD into your CD-ROM drive.
- If the AutoPlay option is turned on, you will see the WoodWorks® Design Office Setup screen. Otherwise,
- From the Windows Explorer, select your CD-ROM drive and double-click on **setup.exe**, or
- Go to the ***Start*** menu and choose ***Run***…
- Type **d:setup** (‘**d**’ being your CD-ROM drive) and press ENTER.
- Follow the instructions on screen.

Downloading Electronic Version

- A link will be provided to purchasers.
- **Complete Registration** information
- **Save** appropriate Design Office .exe file to hard drive or **Run**
- Click on **Downloaded File** and follow **Installation Procedure** instructions.

Downloading Demo Version

- Go to **www.woodworks-software.com**
- Go to **Software Downloads** and select **Begin Download Now**.
- You will be directed through the registration page.
- **Save** appropriate Design Office .exe file to hard drive or **Run**
- Click on **Downloaded File** and follow **Installation Procedure** instructions
- Use "**DEMO**" as the ***keycode***

Installation Procedure

The installation program allows you to specify the installation folder and the start menu folder on your computer. The installation folder is C:\Program Files\WoodWorks\[USA or Cdn]; you need to change this if you wish to also retain a previous version of WoodWorks®.

Retaining Database and Custom Settings
You can retain database customizations made with WoodWorks® Database Editor from an existing installation by specifying the same installation folder as the existing installation, choosing the ***Custom Install*** *Setup Type*, then unchecking ***Custom Materials Database*** in *Select Components*.

You can retain your settings customizations for any of the WoodWorks® programs by specifying the same installation folder as an existing installation, choosing the ***Custom Install*** *Setup Type*, selecting the program name, e.g. **Sizer**, in Select Components, pressing the ***Change*** ... button, then unchecking **Settings** in *Select Sub-components*.

Refer to Read Me files for additional instructions regarding Installing and Uninstalling software.

Network Installations
There is no network install version of WoodWorks®, however the Design Office readme.htm file includes instructions on how to run WoodWorks® from a network (See "Network Installations" in the Readme file).

Registration Keycodes

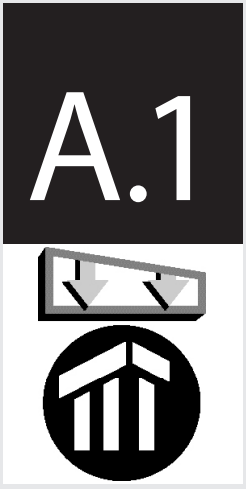
When one of the Design Office programs is run for the first time, a keycode will be requested. Keycodes will only be given to purchasers of the software. Those wishing to simply evaluate the program may type **demo** instead of a keycode.

To receive your keycode, email the Software ID that appears at the bottom of the keycode prompt screen (also found in the "About WoodWorks..." screen) to **sales@woodworks-software.com.**

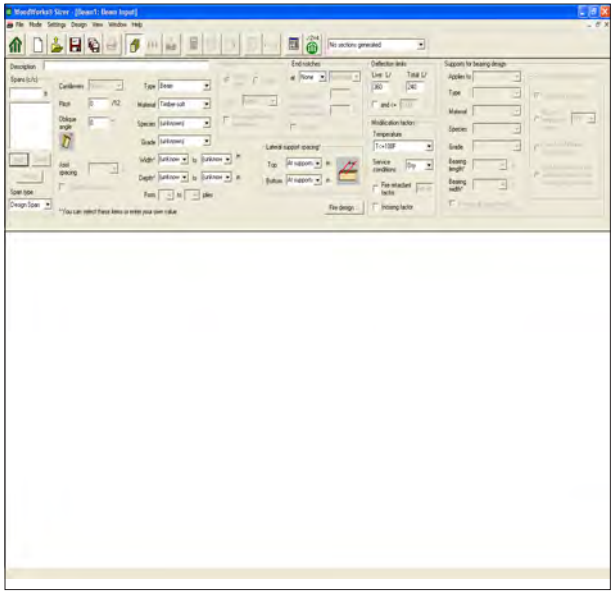
It is very important that you register with your correct email address, fax and phone number(s). This will allow WoodWorks® to send notification of updates and new releases. Please send contact information updates to sales@woodworks-software.com.

Please refer to Read Me notes once the software is installed for topics including Version History, Tech Support, Uninstall.

What is Sizer?



1.1 Operating Modes—Beam, Column & Concept



Note: The Database button on the toolbar allows for viewing of material properties and is accessible in all three modes in Sizer. For more information on the Database Editor see section D.

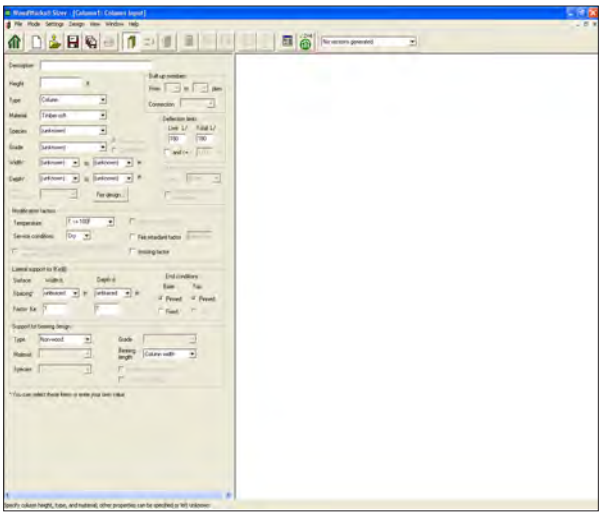
Beam Mode

Beam mode allows you to quickly enter and design individual wood bending members. This includes up to six span continuous beams, or beams with cantilevers. Sizer uses the stiffness method for the analysis.

To select Beam mode when first starting Sizer, click on the ***beam*** toolbar button or select ***Beam*** from the ***Mode*** menu.

The main work area allows you to specify a number of parameters for your beam or joist. The toolbar allows you to quickly change between Beam, Load, Point of Interest, Results and Diagram views and to make Sizer design your beam or joist.

When Beam mode is active, a checkmark is displayed next to ***Beam*** on the ***Mode*** menu.



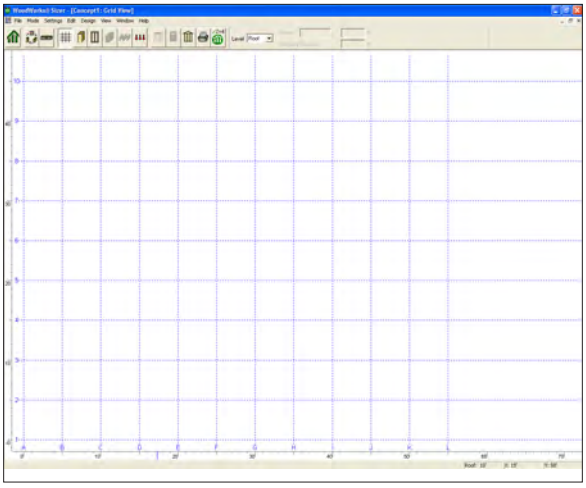
Column Mode

Column mode allows you to quickly enter and design individual wood column or beam-column members. Sizer analyzes using the stiffness method.

To select Column mode when first starting Sizer, click on the ***column*** toolbar button or select ***Column*** from the ***Mode*** menu.

The main work area allows you to specify a number of parameters. The toolbar allows you to quickly change between Column, Load, Point of Interest, Results and Diagram views and to make Sizer design your column or beam-column.

When Column mode is active, a checkmark is displayed next to ***Column*** on the ***Mode*** menu.



Concept Mode

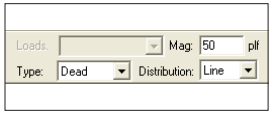
Concept mode is a graphical design and analysis work area for the preliminary design of structures considering gravity loads.

To select Concept mode, click on the ***concept*** toolbar button or select Concept from the Mode menu.

Concept mode provides a graphical work area where you can configure and design a complete structure in plan.

The main work area contains rulers to allow you to position the cursor within the work area accurately and quickly.

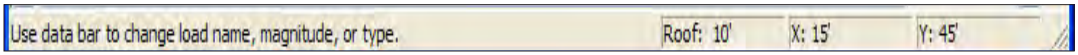
The ***Concept Mode*** toolbar is the box below the menu bar and to the left with fourteen buttons titled ***create or manage project mode, level, grid, column, wall, beam, joist, loads, group, design, elevation view, print, and database***. The ***Concept Mode*** toolbar allows you to change between Grid, Column, Beam, Joist and Load views. The ***run*** button makes Sizer design the elements in your structure and display the results.



The ***Concept Mode Data*** toolbar is the box below the menu bar and to the right of the ***Concept Mode*** toolbar. This toolbar gives a variety of information depending on the selected view. This includes information such as group names, member names, gridpoint elevations, grid line locations, load magnitudes, load locations or joist direction.

The ***status bar*** is located at the bottom of the Sizer window. The status bar indicates the function of a given toolbar button, the current level and height, and the X and Y cursor positions.

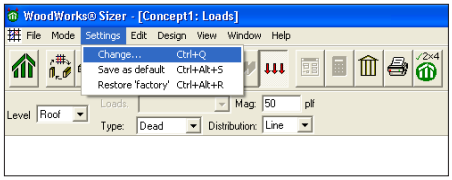
The tool bar, status bar, and data bar are turned on or off through the ***View*** menu. When Concept mode is active, a checkmark is displayed next to ***Concept*** in the ***Mode*** menu.



1.2 Settings

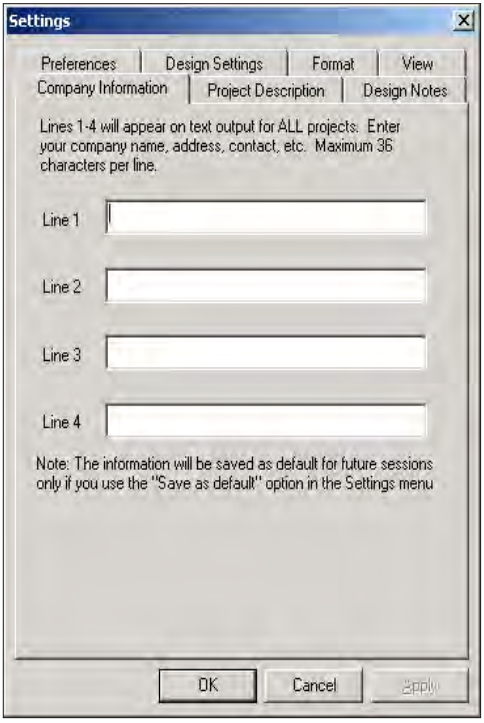
Settings Dialog

Choose ***Settings*** from the main menu and then click ***Change*** to open the Settings dialog.



Company Information Tab

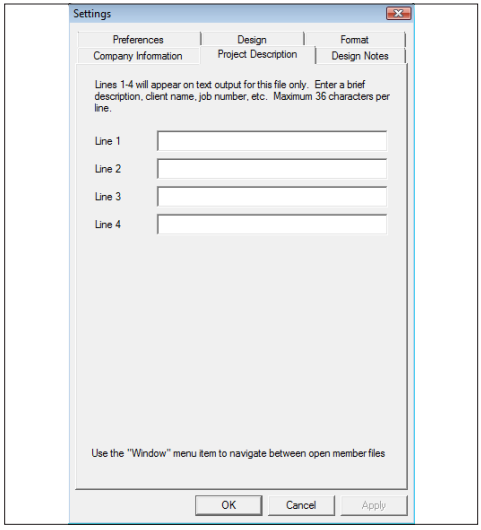
Choose the ***Company Information*** tab from the Settings Dialog. This tab allows you to enter your company contact.



You only need to enter company information once. You can change it at any time by following the procedure described above.

Project Description Tab

Choose the ***Project Description*** tab from the Settings Dialog. This tab allows you to enter project information, including the project name, location, client and job number.

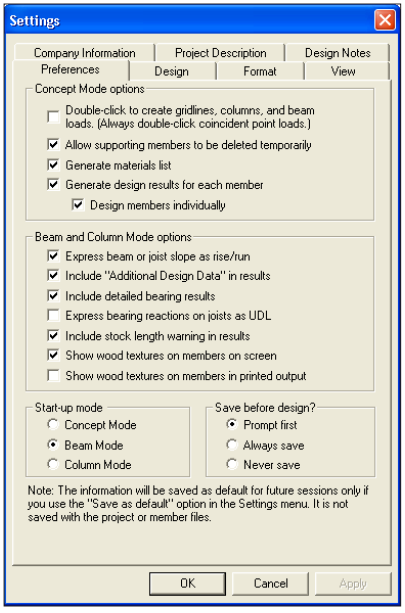


Every time you start a new project, you should enter new project information.

Note: The workspace feature allows for the creation of a "Project" file meant to correspond to a typical user building project, with one concept mode file and a number of beam and column files to be open simultaneously.

Preferences Tab

Choose the ***Preferences*** tab from the Settings Dialog. This tab allows you to make choices about the operation of the program.



Start Up Mode

This item specifies which one of the three design modes—Concept, Beam, or Column— should automatically appear as Sizer begins each session.

Save Before Design

Specify whether or not Sizer should prompt you to save before designing.

Prompt - Sizer prompts you to save the current project prior to starting the design process. This is the default.

Always - Sizer always saves the current project automatically (without prompting) prior to designing.

Never - Sizer will not save the current project prior to starting the design process. There will be no prompt.

Concept Mode Options

These settings apply only to the Concept Mode and allow you to specify:

- whether to use a single or double click with the mouse,
- whether or not supporting members can be deleted temporarily while you make changes to the structure,
- if a materials list should be generated,
- if Design Results should be generated for each member (per group or individually)

Beam and Column Mode Options

These settings apply only to Beam and Column mode and allow you to specify:

- whether to enter beam or joist slopes:
  a) in degrees, or
  b) as a value over 12 (i.e. 4 in 12)

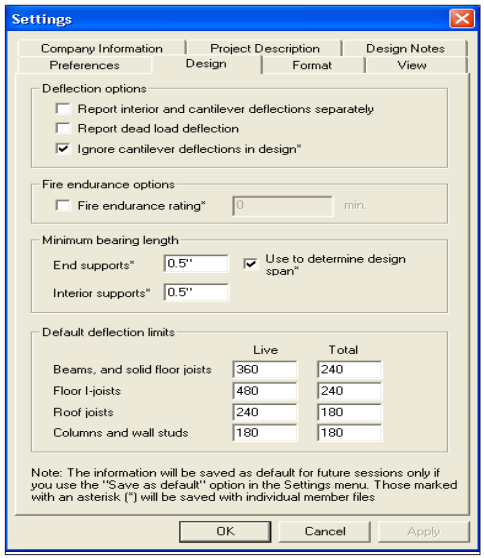
- whether to include the Additional Data section (adjustment factors for reference design values) in the Design Check report,
- if you want to show detailed bearing results in the Design Check report (if not selected, only the bearing length and minimum required bearing length is shown; if selected, additional information including the bearing capacity, the governing load combination, and bearing factors are shown),
- if the joist reactions should be shown as a uniform distributed line load to the support below (ie joist reaction divided by joist spacing), or as the individual total joist reaction (as a beam's reaction would be shown),
- whether to include the stock length warning,
- if the member's wood texture should be shown when viewing the computer screen, and/or when printing.

Design Tab

Choose the Design tab from the settings dialog. This tab allows you to specify deflection options, fire endurance options (USA), and modification factor options (Canada).

Fire Endurance Rating (U.S. Only)
A Fire Endurance Rating option permits you to specify the minimum required fire endurance rating for solid sawn beams and columns.

If you enable this feature, Sizer will add fire endurance as another design



criterion that must be satisfied. The number that appears in the Fire Endurance field is a default that automatically appears for each project. In Concept mode this default can be over-ridden in the Groups dialog on a group-by-group basis. Similar input fields exist in the Fire Resistance and Treatment dialog of Beam and Column modes, that is accessed from the Fire Design button. The number of sides exposed to fire (0, 3 or 4) can also be set.

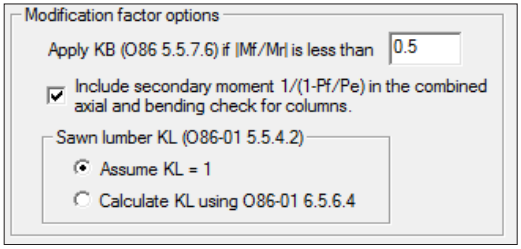
Fire endurance is calculated according to IBC 721.6.3, which is available from the American Wood Council at www.awc.org

Deflection Options
- report interior and cantilever deflection separately in Beam mode's Design Check report,
- report the dead load deflection,
- ignore cantilever deflections in design so that the deflection of a cantilever never governs a design.

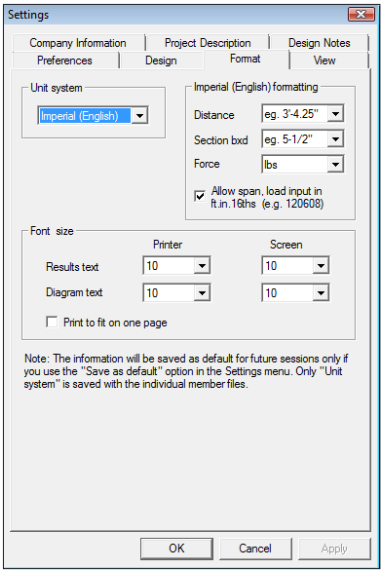
Minimum Bearing Length
Sizer provides designers with the flexibility of setting absolute limits for the minimum bearing length for both exterior and interior support locations. The values entered in these boxes indicate the smallest bearing the program considers. If the program calculates a minimum bearing less than this value, it overrides it with the value entered.

Selecting the *"Use to determine design span"*checkbox uses this absolute bearing length in the determination of beam spans, unless the calculated minimum bearing length is larger. De-selecting this option allows the design span to be based on the calculated minimum bearing length, which will be advantageous if the calculated minimum bearing length is smaller than the user defined absolute minimum bearing length because the design span, calculated from center of bearing, is reduced.

Modification Factor Options (Canada Only)
Apply K_B Factor
Specified strength may be multiplied by the length of bearing factor K_B based on CSA O86 Table 5.5.7.6 provided the conditions in clause 5.5.7.6 are met. The length of bearing factor should not be applied if the points of bearing occur in areas of high bending stress. The software allows the user to specify the maximum ratio of factored bending moment to bending resistance for which the K_B factor will be applied.



For bilateral bending, where a column has an axial load and a load causing bending moment, such as eccentric axial loads or loads perpendicular to the column (like wind), selecting the "*Include secondary moment...*" will amplify the primary bending moment.



K_L Factor Options
When lateral support is provided at points of bearing, the lateral stability factor K_L may be taken as one, provided the maximum depth-to-depth ratio meets the conditions listed in CSA O86 clause 5.5.4.2. Alternatively, K_L may be calculated according to clause 6.5.6.4.

Format Tab

Choose the Format tab from the Settings Dialog. This tab allows you to enter the unit system to be used and format the font size for printing and viewing.

Unit System
This option allows you to select whether Sizer will operate in Imperial (English) or metric units

Imperial Length Format
Choose Imperial (English) units to be displayed in either decimal feet, in feet with decimal inches, or in feet with inch fractions. This can affect either distances or member sizes.

Force is an option that permits the user to select the Imperial (or English) units for Point Loads as either *lbs* (pounds) or *kips*. This only applies to the input fields for loads – the output unit for analysis and design results are always kips.

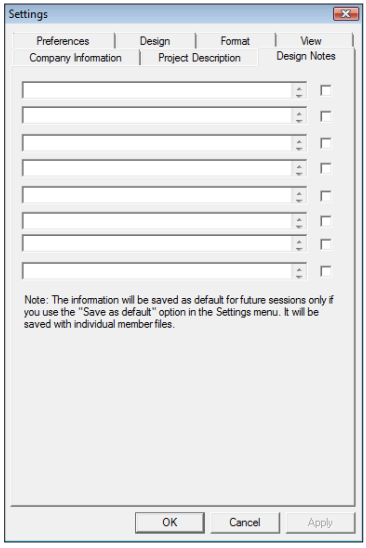
Selecting the "*Allow span load input in ft.in.16ths"* provides designers with an easy way to enter, for example, 12' - 6 1/2" as "120608".

Font Size
This option allows you to set the font size to be used for results and diagram text for either viewing or printing.

Print to fit on one page
This is an option that automatically reduces the printing font to allow the output to fit on a single page. Note that this feature is bypassed if the font size required is less than 4 pt.

Design Notes Tab

Choose the Design Notes tab from the Settings Dialog. This tab allows you to specify standard design notes that are to be added to the Design Notes section of the Design Check report in Beam and Column modes. User specified design notes can be activated or inactivated using the check box to the right of the design note.

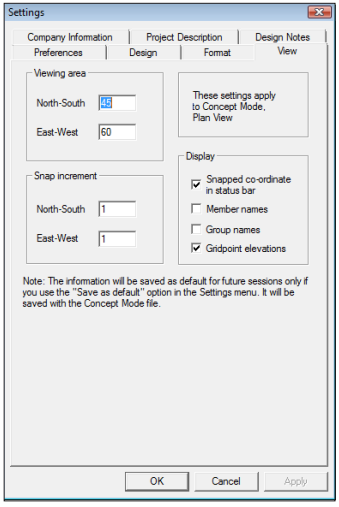


View Tab (Concept Mode only)

Choose the View tab from the Settings Dialog. This tab allows you to specify different viewing options such as the size of the viewing area, the snap increment, the percentage zoom, and member names.

Viewing Area

These fields specify the maximum viewing area in plan for the North-South and East-West directions.



Snap Increment

This specifies the smallest increment a newly created gridpoint will move or "snap" to. For example, a snap increment of 2.0 ft allows you to place gridpoints at 10.0, 12.0 and 14.0 ft, but not at 9.0 or 11.0 ft.

Display

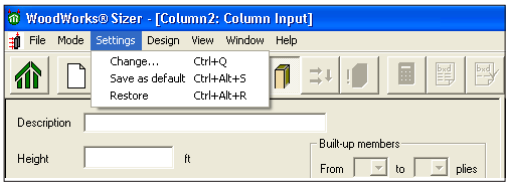
These options allow you to include additional information on the diagrams for viewing or printing. This includes the snapped coordinate, the member names, and the group names. You can also specify whether snapped or actual mouse co-ordinates appear in the status bar.

Save As Default

Choose ***Save new settings*** under the ***Settings*** menu. This will save any new changes that have been done in the Settings menu. These new settings will be the default for any new files that you create. This custom definition will again be in effect during the next Sizer session. However, a previously saved Sizer file with different definitions will over-ride these settings. Clicking ***File*** and then ***New*** will restore your definitions at any time.

Restore Factory Settings

This option restores the original settings that were in effect when Sizer was first installed. They will only be in effect for the current session unless you click on ***Save New Settings***.



1.3 Working with Files and Projects

Creating New Files

When you start Sizer, it displays an untitled screen, allowing you to create a new file.

Whenever you choose ***New File*** from the ***File*** menu, Sizer clears the screen, discarding the current project. If you have modified the current project since you last saved it, Sizer asks you whether to save or discard your modifications.

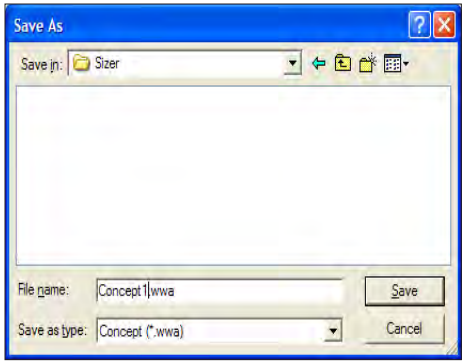
Creating New Projects

In the file menu, there is an option of creating a new project (this can also be done by clicking on the ***Create or Manage Projects*** button on the toolbar), which allows several files to be grouped together from Sizer to form a project. When creating a new project, Sizer will prompt you to enter a project description that will be shared by each file in the project. The project can contain only one concept mode file and a number of beam and column files.

Saving Files

If you choose ***Save File*** from the ***File*** menu, Sizer saves the current file using the last name you gave it. If the file is a new one and you have not given it a name, the ***Save File*** command has the same effect as ***Save File As...*** (see below).

If you choose ***Save File As...*** from the ***File*** menu, the current file is saved with the name you enter. In the Save As dialog box, the File Name field contains (by default) the last



name given to the file. You can specify a different name if you wish.

To change the file type, click the down arrow, select a file type and double-click it.

To change folders, select a folder from the Folders list and simply double-click the folder.

Click on ***Save File As...*** to save the file under the new name. If a file already exists with the name you specified, Sizer asks you to confirm that the existing file should be overwritten. Click on Cancel to exit the ***Save File As...*** dialog without saving your file.

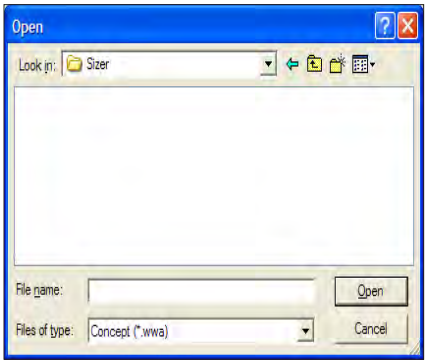
If you choose ***Save Project*** from the File menu, Sizer saves the current files in the project with the last name you gave it. If the project is new and or has not been given a name, the ***Save Project*** command has the same effect as ***Save Project As...***

If you have several files open (beam, column and a concept) and wish to save all the open files into a new project, simply click on the ***Create or Manage Project*** button on the toolbar.

The ***Save All*** button has the same effect as ***Save*** for all the open files.

Opening Files

To open an existing project, choose ***Open File*** from the ***File*** menu. The Open dialog appears. Type a file name (or select one from the list) and then click ***Open*** to open that file.



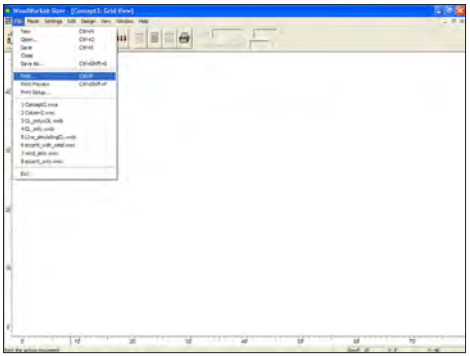
If you select a new folder in the Folders list, the selected folder becomes the current folder.
Select ***Cancel*** to return to editing the current file without opening a new one.

Opening Projects

To open an existing project, select *Open Project* from the *File* menu. The open dialog appears and the file type is .wprj. When a project is opened, all the files within that project are opened.

Printing Files and Graphics

To print a file or on-screen graphics, click on the ***Print*** button from the toolbar or choose ***Print*** from the



File menu. This will print the text or graphics image that is currently shown on screen.
Depending on the mode you are in, you will have the choice of printing various text and graphics files as described below.

Concept Mode

Results by Group

This text file contains the suggested sections for each group in a given structure after Sizer has designed them. This is the default design output when Sizer performs a design.

Results By Members
The Results by Member output contains design results by group or by individual member, depending on the setting used in the Settings Preferences tab. In both cases, Sizer identifies how each individual member has performed in the design stage.

Materials List
This file contains a materials list, per floor, of all the framing elements used in your design.

Graphics
The Concept mode allows you to print the plan and elevation views of the structure as they are displayed on the screen.

Beam and Column Modes

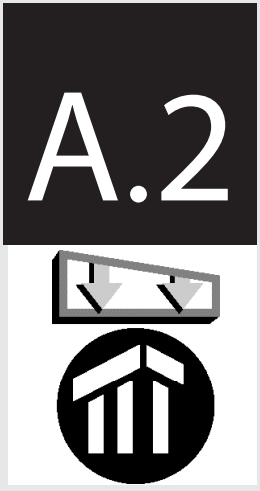
Analysis Results Text
This file contains the analysis results computed by Sizer. This includes such things as the maximum shears and bending moments.

Design Results Text
This file contains the suggested sections for a given beam or column after Sizer has designed it.

Design Check Text
This file contains the design results computed by Sizer for a specific section. It includes such things as allowable shear and bending stresses.

Graphics
The Beam and Column modes allow you to print the reaction, shear, bending moment, and deflection ***diagrams.***

Designing a Beam



2.1 Beam Mode

What It Does

Beam mode allows you to quickly enter and design individual wood bending members. Beam mode provides extensive information about a single beam or joist in your project. Typically you would use Beam mode to verify the design of critical members.

To select Beam mode, choose ***New File*** from the ***File*** menu and select ***Beam Mode*** or click on the ***Beam*** button on the toolbar.

When Beam mode is active, a checkmark is displayed next to ***Beam*** on the ***Mode menu***.

Transfer of a Beam or Joist from Concept Mode
If you are operating in Concept mode and you wish to further analyze a particular beam or joist, click on the beam or joist to select it and then choose ***Beam*** from the ***Mode*** menu or click the ***mode*** button on the toolbar.

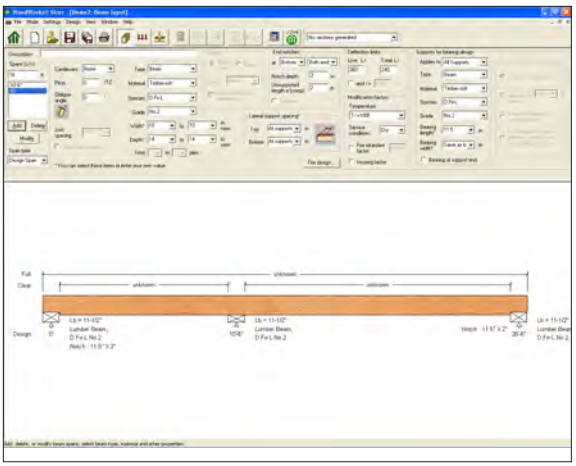
Sizer first determines the design loads for the member in question, based on the structure entered in Concept mode. Sizer automatically transfers the load information to Beam mode's input fields.

Note: The transfer from Concept to Beam mode is one-way. Any changes made to the transferred member cannot be exported back to the original member in Concept mode. There are a few exceptions, such as the selection of **Self Weight**: "manually input " or "automatically included in loads analysis". Refer to **Help** on "*Apply options to Concept Mode*" which refers to what options can be tranferred with a click of a button in the ***Loads*** view, activated if Concept mode is open.

Transfer of Beam or Column from Autodesk's Revit® to Sizer
If you have a structure designed in Revit®, you can import it into Sizer for specific design verifications. For more information, visit our website at www.woodworks-software.com.

2.2 Beam (or Joist) Parameters

Click the ***beam*** button on the toolbar to describe the beam (or joist) to be designed.



Sizer displays a diagram of the spans in the lower portion of the screen. The Design Span, Clear span, and Full span are all shown on the diagram. Sizer can also display cantilevered and sloped beams.

Description
A description of the beam can be entered which will appear in the *Design Results* output and *Design Check* output.

Spans and Span Type
This list box defines (in left-to-right sequence) the length of each span and cantilevers.

Before entering spans it is important to understand what type of span is being entered. Immediately below the *Add, Delete* and *Modify* is a "Span type" entry where Design, Full, or Clear span can be selected. "Design span" has been the traditional method used to enter spans in Sizer, and is the distance between support points on beams. Support points for exterior supports are ½ the minimum required bearing length from the inside of the support. For interior supports, they are at the centre of the support. The clear span is the distance between the inside faces of the supports, for all spans. Full span is measured from the outside faces of exterior supports, to the centre of interior supports. It is the beam length for single-span members. The graphical representation of the beam accurately illustrates these span types using dimensions.

To change the length of the span, select the span in the list of spans, change the length in the spans box, and then click ***Modify.***

To delete spans, click the span you wish to delete and then click ***Delete***.

Cantilever
This drop-down list is only active when two or more spans have been added and it specifies cantilevers at either end of the beam. The outer end of a cantilevered span is unsupported. The default is None (no cantilever at either end).

Slope/Pitch
This field specifies the slope of the beam in degrees or as a pitch. A horizontal beam (the default) has a slope of zero. A beam that slopes up

to the right has a positive value. A beam that slopes down to the right has a negative value.

Level bearing is assumed and thus there is no horizontal thrust at supports. Sloped members are only designed for flexure. It is assumed that axial force is insignificant in these members.

Joists supported by sloped members are spaced with respect to the longitudinal axis of the sloped member.

Built-up Members
If the selected material can be used in built-up or multi-ply sections, these two drop-down lists specify the range of plies to use. The default selection is *(unknown)*, which forces Sizer to select suitable sections for a range of plies.

If the material cannot be used in multiply sections, these lists are disabled.

Type
This drop-down list specifies the type of bending member to be designed - beam, floor joist or roof joist. The default selection is **Beam**.

Material
This drop-down list specifies the material database to use. Standard choices include lumber, lumber n-ply, timber, glulam and many others. The default for beams is **Timber.**

Species
This drop-down list specifies the species of wood to use for member design. Available species depend on the selected material.

The default selection is (*unknown*), which forces Sizer to select suitable sections from each species.

Grade or Combination
This drop-down list specifies the grades of wood to use to design the member. Available grades depend on the selected database.

The default selection is (*unknown*), which forces Sizer to select suitable sections from each grade.

This list box is unavailable if the species is specified as (*unknown*).

Width and Depth
The two width drop-down lists specify the width range of the sections Sizer should use when designing the member. Similarly, the two depth drop-down lists specify the depth range. Available widths and depths depend on the selected database, species and grade.

The default selection for each of these drop-down lists is (*unknown*), which forces Sizer to select from a full range of section sizes.

You can limit the section sizes to be considered by selecting appropriate width and depth ranges in the drop-down lists.

These drop-down lists are unavailable if the grade is (*unknown*).

To specify a custom section, enter some non-standard size (in actual dimensions) in each drop-down list. Note that a custom size may not be commercially available.

Note: Sizer performs a ***design check*** rather than a ***design selection*** if both the width range and depth range specify single values. Sizer cannot design a section if only the width or depth is a custom size. For example, you cannot have one field as *(unknown)* when the other contains a custom size.

Deflection Limits
These fields allow you to specify the deflection limits to be used for design, based on the span (example L/360) and in absolute terms (1 inch). Default deflection limits are entered in Settings [Design] tab. See also the "Loads" button and "Load types and combinations" for long term deflection (creep) and load reductions (US only).

Fire-retardant Factor
This field permits you to specify the reduction in bending, shear, and deflection resistance of wood treated with fire-retardant chemicals. In the U.S. version, to specify a value, click the check box.

Notches
Sizer only designs notches at the ends of a beam. Notches can be specified as being on the **top** or **bottom** of a beam. Both notch depth and length are required as input.

Lateral Support Spacing
These drop-down lists permit top and bottom lateral support to be specified as ***Full*** (full lateral support), ***At Supports*** (lateral support provided at bearing supports only), or at a specified numeric interval. The defaults are ***Full*** for the top and ***At Supports*** for the bottom.

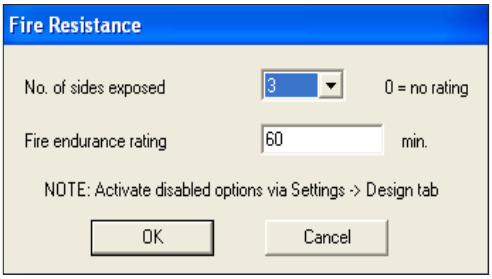
Note: For some materials, this selection is disabled. Notes in the output specify the necessary lateral support.

Glulam Lay-up
This field allows the input of wane for certain combinations, with associate strength reduction. It also allows the input of non-edge bonding with associated strength reduction

Oblique Angle
This field specifies the angle for oblique purlins. The default is blank (no angle). By specifying an angle of 90°, you can investigate the use of the selected material database as a plank.

Treatment (Canada Only)
When you select Fire-retardant treatment or Preservative treatment from the drop-down, an input field becomes active for you to input the corresponding modification factor.

Fire Design Button (U.S. Only)
Clicking on this button opens the Fire Resistance dialog that allows you to specify fire endurance design criteria provided the Fire-endurance rating is activated in the Settings / Design tab.



Fire Endurance Rating
Sizer checks the Fire Endurance Rating of timber and glulam beams or columns. Sizer calculates the *Fire Endurance Rating* based on the number of exposed sides and on the loads applied to a member. To activate the fire endurance check, the number of exposed sides must be defined. If the number of exposed sides is set to zero, the program will not perform the fire endurance check.

Fire Endurance Rating is activated through the Settings/Design tab.

Joist Spacing
Active for joists only, this drop-down list specifies the joist spacing. You can select one of the standard 12 in. (300mm), 16 in. (400mm), or 24 in. (600mm) spacings or enter your own value.

The tributary width for uniform area and partial area loads in Loads view is equal to the joist spacing set in the Joist Spacing field.

Incising Factor (U.S. Only)
This field allows you to specify whether an incising modification factor should be applied according to NDS 4.3.8.

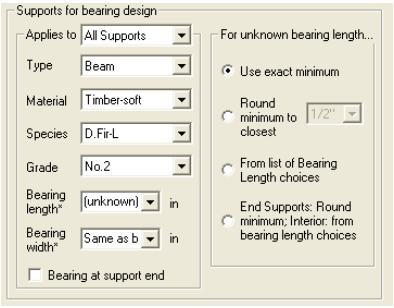
Repetitive Member Factor (U.S.)/System Factor (Canada)
This control allows you to specify if a load sharing factor is to be applied.

Service Conditions
This drop-down list specifies either *Wet* or *Dry* service conditions for the beam. The default is ***Dry***.

Temperature (U.S. Only)
The default is T<100°F. Other values for T are 100°F<T<125°F and 125°F<T<150°F.

Supports for Bearing Design
These fields allow you to select the type of bearing as well as the support material for the bearing design. The type can be selected as hanger, beam, column, sill plate or other non-wood. The bearing length or width can be entered or the calculation of the bearing length can be made more specific.

The bearing design information can be entered for "all supports" at once



(as shown in screen capture) or separately for "interior" versus "exterior" supports, or for each support independently. When the bearing length is not entered (or left as "unknown"), the "For unknown bearing length" entry is activated, and several choices are made available including: using the exact calculated minimum bearing length, rounding the calculated bearing length to the closest user specified designation, automatically rounding up to the next choice in the "Bearing Length" dropdown box, or a combination of the previous two options useful for when a beam sits only partially on end supports but fully on interior supports.

Minimum bearing lengths can be specified in the ***Design*** tab of the ***Settings*** menu.

Vibration Buttons (Canada Only, Floor Joists Only)

This button opens the Vibration Design dialog box that allows you to specify details that affect floor vibration. Vibration analysis is carried out according to A-9.23.4.2.(2), Appendix A of the National Building Code of Canada.

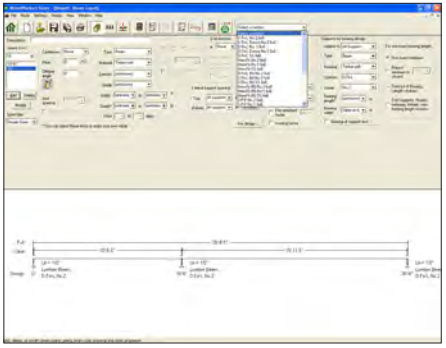
Selecting Sections for Design Check

Sizer provides you with an extensive list of suggested sections for the beam it is designing. To get more details about a single section size, you can perform a design check on that section.

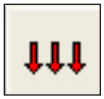
After running an initial design, select the section you want to check from the drop-down list entitled ***Suggested Beam Sections*** at the right side of the toolbar.

Sizer automatically fills in the Species, Grade, Width and Depth fields with the appropriate values. Sizer automatically performs a design check of the selected section.

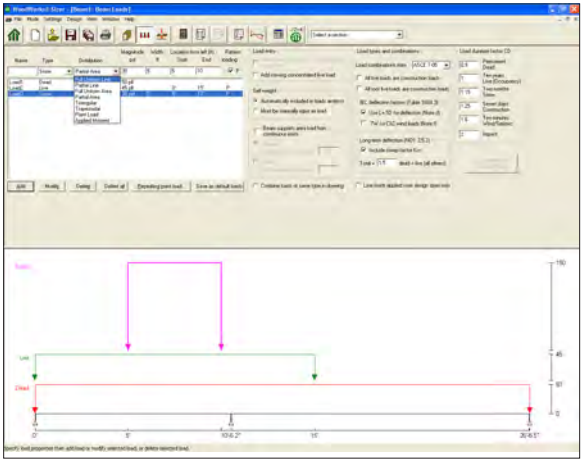
The list of suggested sections on the toolbar remains available until you perform another design (rather than a design check).

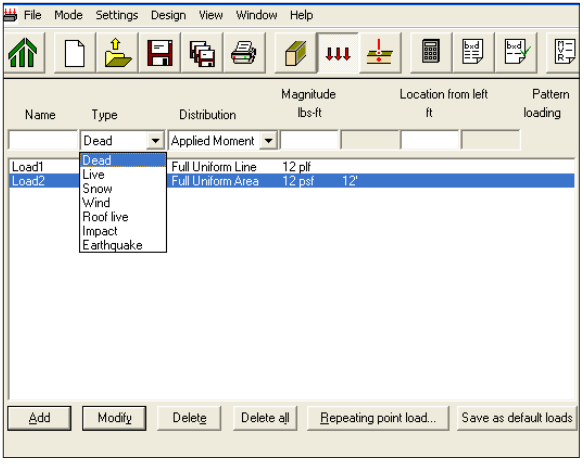


2.3 Loads



Click the ***Loads View*** button on the toolbar to specify the loading of a beam (or joist) that is to be designed. The Loads view opens.





Name
Use the Name field to enter the name of the load you want to apply to the beam. (Sizer generates an appropriate name if you leave this field blank.) The list box beneath this field contains the names of all the loads you have specified for this beam. Click one to select it.

Type
This drop-down list specifies the type of load being applied. Sizer allows you to select from a variety of load types, including dead, live, roof live (U.S. only), snow, wind, impact (U.S. only), sustained live (storage and contained fluids) and dead (soil) (Canadian only), earthquake, and hydrostatic (Canada only, columns only). The U.S. version also allows all live loads or all roof live loads to be considered a "construction" load, and will use the appropriate duration factor if this is selected.

Depending on the load type specified, Sizer will apply the correct load duration factor and load combination factor to each load combination. For the Canadian version, Sizer also applies the correct load factor. The default load type is ***Dead***.

Distribution
This drop-down list specifies the type of load distribution: Full Uniform Line, Full Uniform Area, Partial Line, Partial Area, Triangular, Trapezoidal, Point and Applied Moment. At the bottom of the input section is a "Line loads applied over design span only" checkbox. If checked the software distributes the line and area loads to the design span, not the entire beam. This is more a tool to compare files made in previous versions which only applied loads to the design span.

Magnitude
This field specifies the magnitude(s) of the loads being applied. These magnitudes should be entered as specified loads (such as those stated in building codes). When using English units, Sizer allows you to change whether point loads are entered in pounds (lbs) or kilopounds (kips) through the Settings Format tab.

Note that area loads are converted to and displayed as line loads (plf or kN/m) on the load diagram.

Magnitude: Width
This field is only active for Full Uniform Area and Partial Area load distributions and is equal to the tributary width for the bending member. For joists, this value automatically corresponds to the joist spacing selected in Beam input view.

Location From Left
This field specifies the location of the selected load. It is active if the load distribution is Partial Line, Partial Area, Triangular, Trapezoidal, Point Load and Applied Moment.

For Partial Line loads, specify a start and an end measurement. For Triangular loads, specify a location where the load is zero and a location for the maximum load. For Trapezoidal loads, specify the locations of the minimum and maximum loads.

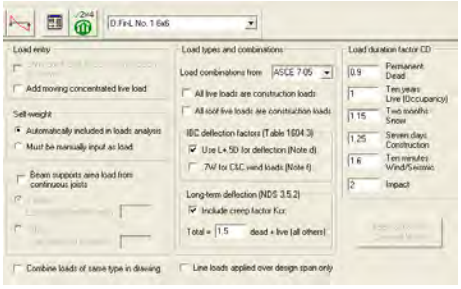
Pattern Loading
Check this box to activate automatic pattern loading. Pattern loading is available in the Beam mode and is applicable to multi-span beams or joists when a live or snow load type is being applied continuously across the member. Sizer will take the live load and pattern it on the various spans to find the worst case for design. Pattern loading is an option that can be turned on or off at the Load input stage. Live loads are patterned as full- or no-load intensity. Snow loads are patterned as full- or half-load intensity.

Save as default loads:
Once loads have been entered, clicking the "*Save as default loads*" button stores the loads so that the next time the same member type is selected, the default loads will appear.

Saving default loads deletes the default loads that were previously saved for a member type. You can eliminate default loads by pressing ***Save*** as default loads for a blank set of loads.

If repeating loads are included, the program includes only those repeating loads that were applied to the original member when default loads are saved. It is advisable to create the longest possible member to apply default repeating loads to.

Default loads are automatically created for members that have not been saved to disk (and do not have a file location). Once a member has been saved, no loads will be automatically created or deleted.



Repeating Point Loads
Click this button to specify multiple point loads with equal spacing and magnitude.

Loads

Load Direction Assumptions
Sizer automatically applies the loads according to the following rules:

- Most loads, including snow, live and dead loads, are applied vertically.
- Wind loads are applied perpendicular to both sloped and horizontal members.

Gravity loads (downward) are represented by a positive load magnitude while suction (upward) loads are represented by a negative load magnitude.

Sloped Member Load Location
When specifying the location of a load on a sloped member, use the horizontal projected length as a reference.

Sizer automatically applies dead and wind loads along the actual length of a sloped member. All other loads are applied along the horizontal projected length of a sloped member.

Load Entry
"***Enter point load as UDL***" uses the spacing to convert a point load to a uniformly distributed load on joists based on the spacing. More typically, this is used to create point loads on joists based on a uniformly distributed load, say from a wall above. This is especially useful when the spacing of joists is being reviewed, in that the line load from above is automatically converted to the correct point load on the joist based on revised spacing. For example a 1200 plf line load from a wall perpendicular to the floor joists imposes a 2400 lb point load on joists spaced at 24" o.c., 1600 lbs per joist when spaced at 16" o.c., and 1200 lbs per joist when spaced at 12" o.c. Instead of manually revising this point load for each spacing run, this load entry tool automates the process.

Add moving concentrated live load (Floor Joists Only)
Only applicable to floor joists, this option allows you to design a floor joist for a moving concentrated floor live load that is required by

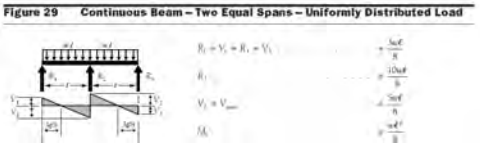
most building codes for certain building categories or floor uses. When selected, Sizer automatically creates a concentrated live load with a default magnitude that acts over a default width. The magnitude and width can be modified. During the design process, Sizer will determine the worst affects of the concentrated load located anywhere along the length of the member in conjunction with all other applied loads, except live loads, and all required load combinations where the concentrated live load acts in place of any loads of type live specified by the user.

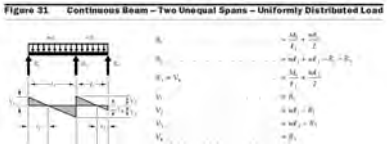
Self-Weight

If *"automatically included in loads analysis*" is selected by the user, the self-weight of a member is considered in the design. The actual effects of self-weight are calculated by multiplying the specific gravity of the material times the area of the section. This load is then added to the dead load when checked. Alternatively the self weight can be ignored or manually added by selecting "*Must be manually input as load*"

Beam supports area load from continuous joists

When continuous joists, rather than simply supported joists, are loading on a beam, the reaction from the continuous joist is greater than the reaction that would be imposed simply supported joists that end at the beam. In other words the reactions the continuous member imposes are not based on the tributary area alone. For example, when a continuous joist spans over the supporting beam, and the joist length is equal on both sides of the designed beam, the reaction imposed on the designed beam is 1.25 times greater than would be calculated by the tributary area alone, as per below Figure 29 from AWC's *Design Aid No. 6, Beam Design Formulas with Shear and Moment Diagrams*. The general case where the joists are continuous over the designed beam with unequal spans is also shown below, from *Design Aid No. 6,* Figure 31.





Accounting for the increased load a beam should be designed for when joists are continuous over the beam is done by selecting the "*Beam supports area load from continuous supports*" checkbox, and either indicating the ratio of the joist span on either side of the beam being designed by first selecting "*2-spans*", or indicate the percent of load on the beam manually by selecting "*Other*".

"*Beam supports area load from continuous supports*" is only enabled when a beam is being designed, and only applies to full uniform area and partial area loads.

Combine loads of same type in drawing
This option only applies to the loading diagram. When enabled, individual loads are accumulated into a single loading profile for each load type (dead, live, etc). When not enabled, loads overlap with other loads of the same type. The selected load appears in bold.

Load Types and Combinations
Sizer combines loads based on the Allowable Stress Design method using the basic load combinations in the IBC/ASCE (US) and Limits States Design load combinations (Canada). In addition to considering the building code load combinations based on the load types present, Sizer also creates load combinations to consider the affects of pattern loading live type loads for multi-span beams or joists when pattern loading is active. Refer to the Pattern Loading section for more information on pattern loads. The designer should verify that the load combinations used are adequate.

For Beam mode, Sizer outputs a list of load combinations with an explanation of each load combination number referred to in the results along with the appropriate load duration factor.

When snow loads are present S and s represent full and half snow loads respectively (e.g., pattern: SsS). When both snow and live loads are present X represents the full application of both types (e.g., pattern: XsX - U.S. Only). Wind loads are not patterned since they are assumed to apply to all spans simultaneously.

For more information on load combinations or on pattern loading, refer to Online help. You can check obscure load combinations by manually adjusting the load duration factors (U.S. only), or performing individual load combinations and load patterns.

Deflection Factors (U.S. Only)
There are two leniencies offered in the IBC related to deflection. These optional IBC deflection factors can be selected using two checkboxes. The first allows the deflection resulting from L + D to be calculated using L+ 0.5D for wood structural members having a moisture content under 16 percent at the time of installation and used in dry conditions. The second option is related to components and cladding wind loads, where deflection may be calculated using only 70 percent of the wind load.

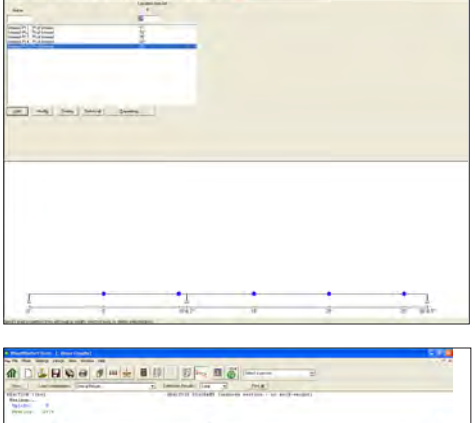
The NDS provides guidance on considering the effects of long term deflection by increasing the contribution of dead load by a creep factor, Kcr, that typically ranges from 1.5 to 2; any value can be entered if the "*Include creep factor*" is checked.

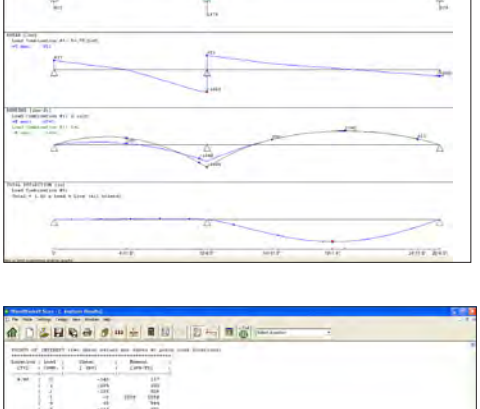
Load Duration Factors
Sizer applies load duration factors according to the load type. These factors are set to the standard NDS (U.S version) or CSA O86 (Canadian version) values by default but these can be changed using this form. (Note that in the Canadian version, when D > L, Sizer determines the default duration factor for standard term loads according to the equation shown in CSA O86-01 4.3.2.3 - refer to the online help for details).

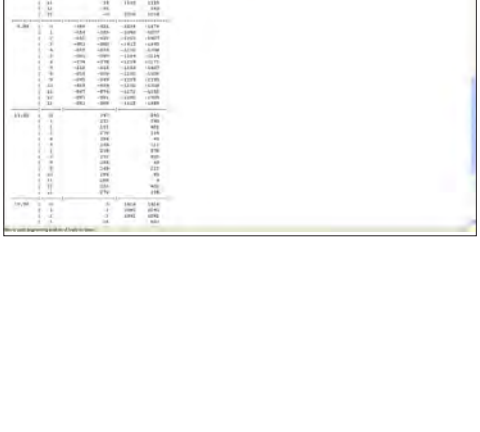
Apply Options to Concept Mode
This button in Beam Load View and Column Load View, is only active when there is a Concept mode file open, and allows the application of the following settings to have an affect on Concept mode. This items can only be accessed from Beam or Column mode, and are:

- Self-weight
- Line Loads applied over design span only
- Long-term deflection factor
- Use L+0.5D for deflection
- Load duration factors CD
- Load combinations from…
- Temperature (from beam view)

2.4 Points of Interest







Click the ***points of interest*** button on the toolbar to investigate the shear and moment at any point along the length of a beam or column. A point of interest is generated by specifying a location to perform the analysis. Now click ***Add*** to add this to the list. Several points of interest can be specified.

After performing a design, the point of interest results will be shown in the Diagrams window and in the Analysis results output.

2.5 The Design Process

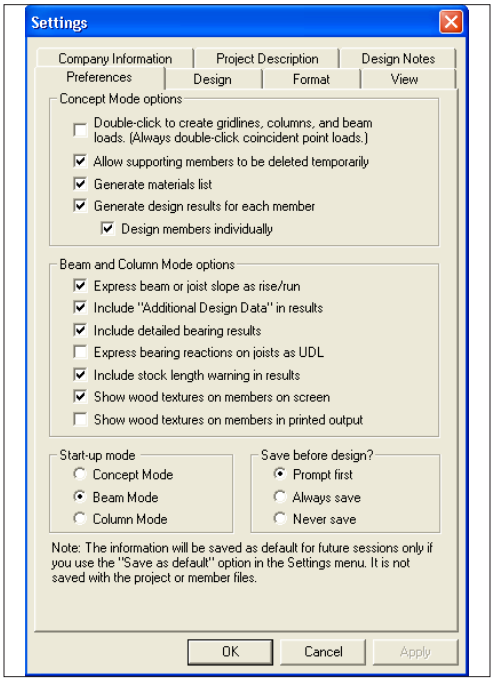
Starting the Design Process

To start the design process, click the ***design*** button on the toolbar. Sizer performs an analysis, designs your beam with the information you entered and automatically displays the results.

If you selected *(unknown)* for some entries, Sizer selects a series of suitable sections that provide acceptable results. If there are no *(unknowns)* and you have not specified ranges in the width or depth fields, Sizer does a design check on the specified section and verifies that the results are within the selected design code's limits.

The number of sections that Sizer selects depends on how many fields you specified as *(unknown)* in the beam screen.

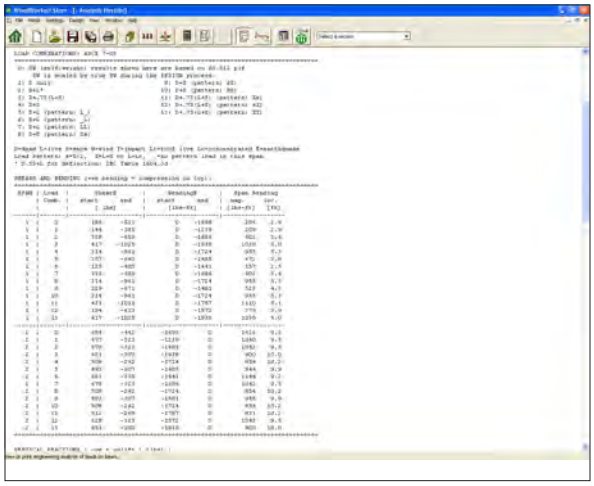
Sizer normally prompts you to save the current project prior to doing the design. To change this, choose one of the ***Save Before Design*** options from the ***Settings*** dialog.

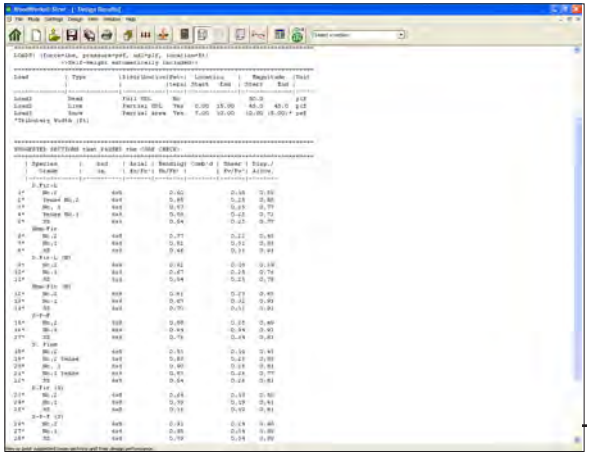


Save Before Design (Prompt first) Tells Sizer to prompt you to save the current worksheet prior to starting the design process. This is the default.

Save Before Design (Always save) Tells Sizer always to save the current project automatically (without prompting you) prior to starting the design process.

Save Before Design (Never save) Tells Sizer not to save the current project prior to starting the design process. Sizer will not even prompt you to save.





Viewing and Printing the Results

Once Sizer has designed the beam, it creates results files. Sizer then gives you several options for viewing the results.

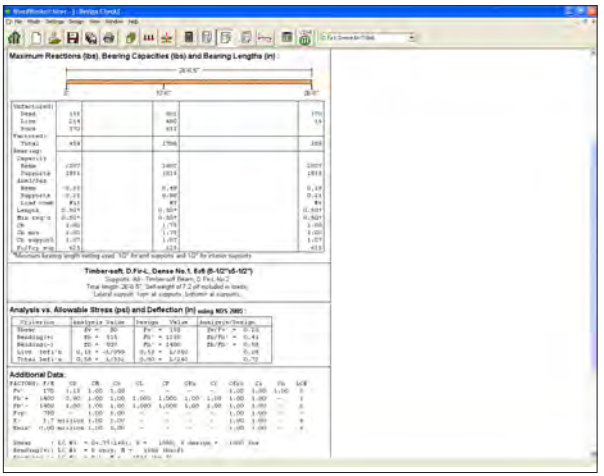
Analysis Results

Click the ***analysis*** button on the toolbar to see the analysis results for each load combination (maximum shears, bending moments and so on) for your beam or joist in tabular form.

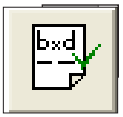
Design Results

If you specified any field as *(unknown)* for your beam or joist, Sizer computes all the possible beam or joist sections during the design process. It also computes the ratios of analysis/design values for bending, shear, and displacement for each section.

To see these results, click on the ***Design Summary*** button on the toolbar menu or choose ***Design Results*** from the ***View*** menu.



Design Check

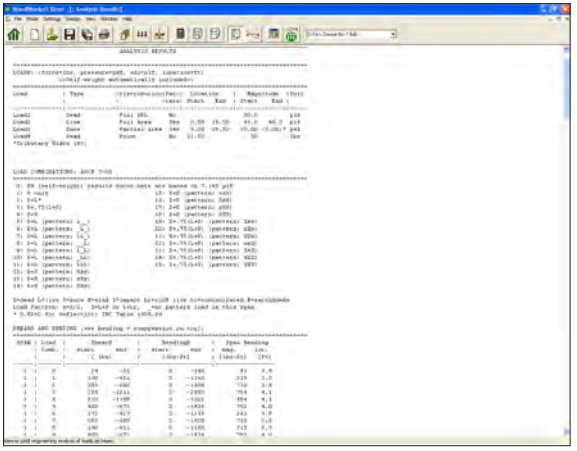


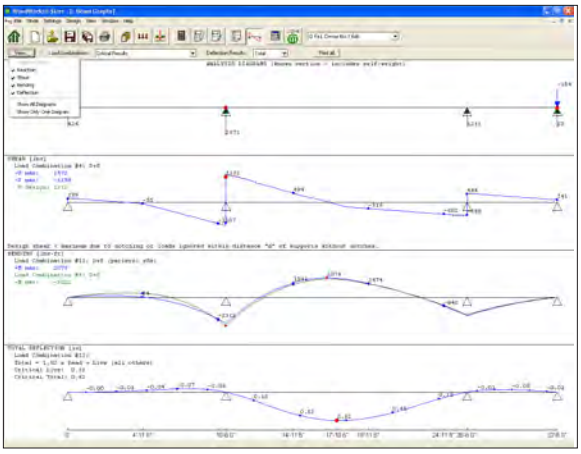
If you requested a specific beam or joist size, Sizer performs a design check and computes analysis and design values (for example shear and bending) for that section.

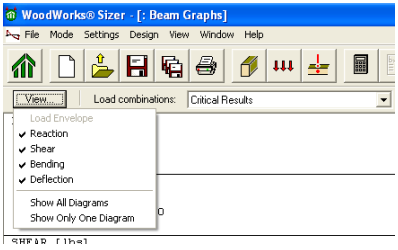
To see these results, none of the parameters can be left as *(unknown)*. Then click ***check*** on the toolbar.

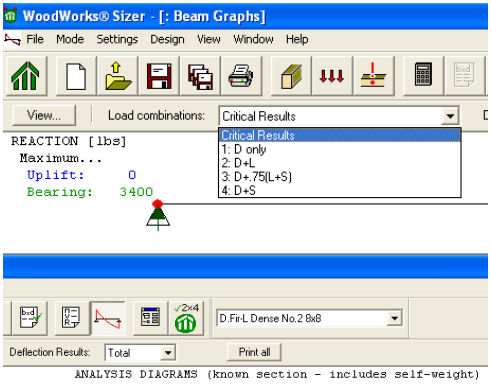
Additional Data Output:

The Design Check output for beams and columns also outputs an Additional Data section that describes the load combination that governed the design in each of the cases for bending, shear and deflection as well as modification / adjustment factors and shows the governing load combination.

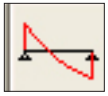








Analysis Diagrams



Click the ***diagram*** button on the toolbar to view the analysis diagrams.

Sizer creates four analysis diagrams including Support, Shear, Bending, and Deflection diagrams.

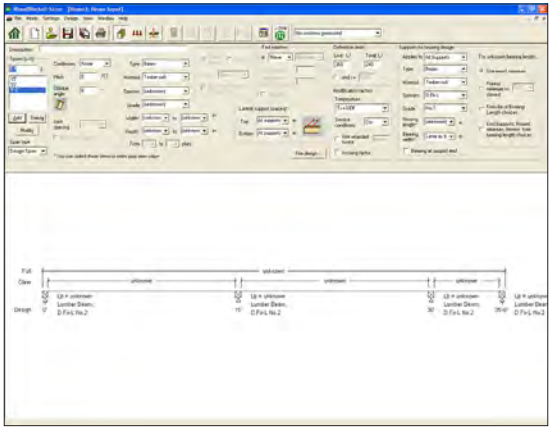
To print the current diagram, click the ***print*** button on the toolbar.

- ***View…*** – This button controls which analysis diagrams are shown, including the Load Envelope, Reaction, Shear, Moment and Deflection diagrams. The Load Envelope diagram is not available when Critical Results is selected as the load combination.
- ***Load Combinations*** – The diagram view results shown are based on the load combination selected in this pull-down menu. When Critical Results is selected, the analysis diagrams shown are for the worst case results of all load combinations. Load combination numbers shown correspond to those used throughout Sizer (Analysis Results, Design Check, diagrams, etc.).
- ***Deflection Results*** - The deflection results can be viewed as total load or live load, based on this pull-down menu.

Printing Results

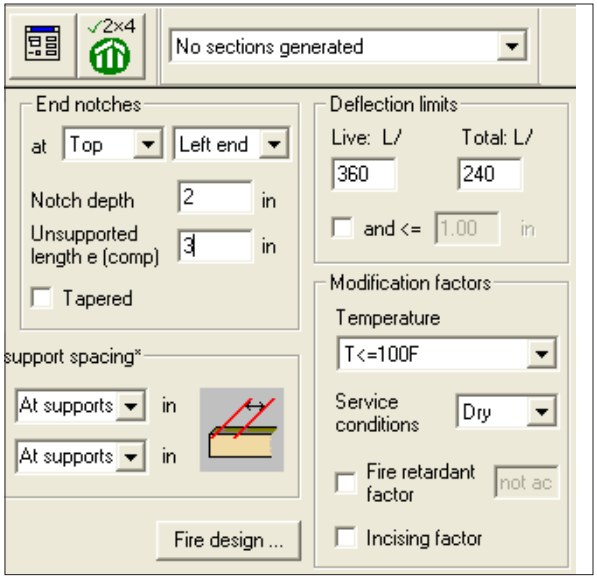
To print results, click on the ***Print All*** button from the toolbar when the result file or graphics screen that you wish to print is on screen. To batch print the analysis diagrams for all load combinations, press ***Print All***.

2.6 Tutorial

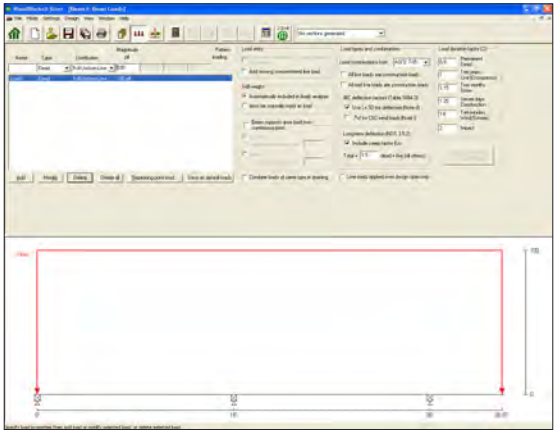


Defining the Parameters

1. Start the program in Beam mode.
2. Select the ***Span*** field and enter a span of 15 (ft).
3. Click ***Add***.
4. Repeat steps 2 and 3 for two additional spans of 15 ft and 5 ft 6 in.
5. Choose ***Right*** from the ***Cantilever*** drop-down list.
6. Although you can try different materials, select ***Lumber n-ply*** for this example.
7. Select species type ***S-P-F***.

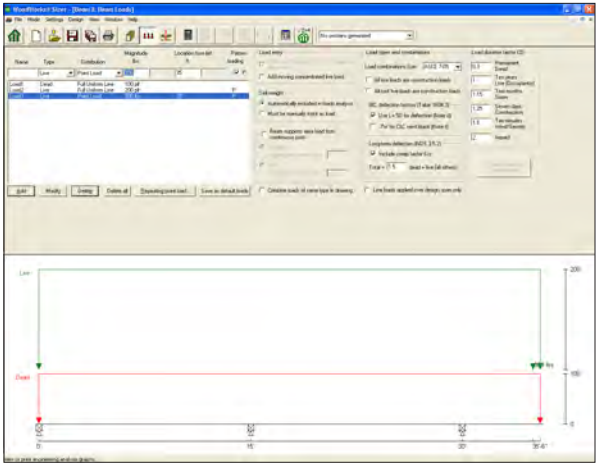


8. Under ***End Notches*** select the at ***Top*** field and enter a depth of 2 (in) and an unsupported length e of 3 (in) .

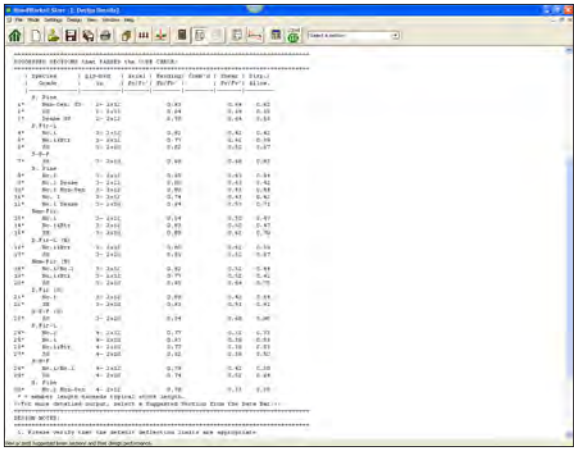


Loading the Beam

1. Click the ***loads*** button on the toolbar.
2. Choose ***Dead*** from the ***Type*** drop-down list.
3. Ensuring that the ***Distribution*** field is ***Full Uniform Line***, select the ***Magnitude*** field and enter a value of 100 (plf).
4. Click ***Add***.



5. Choose ***Live*** from the ***Type*** drop-down list.
6. Select the ***Magnitude*** field and enter a magnitude of 200 (plf).
7. Ensure that the ***Pattern Loading*** box is checked.
8. Click ***Add***.
9. Changing the ***Distribution*** field to ***Point Load***, select the ***Magnitude*** field and enter a magnitude of 350 (lbs).
10. Select the ***Location From Left*** field and enter a distance of 35 ft 6 in.
11. Choose ***Live*** from the ***Type*** drop-down list and click the ***Pattern Loading*** checkbox.
12. Click ***Add***.



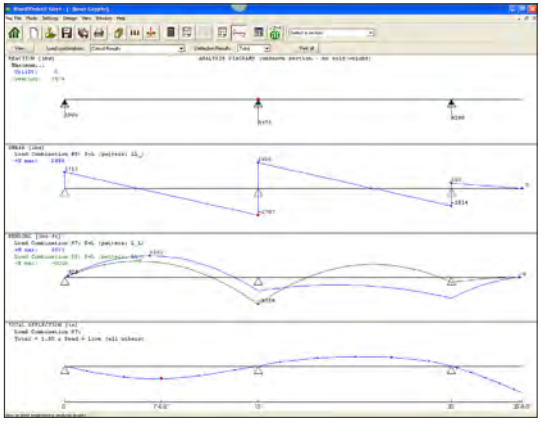
Designing the Beam

1. Click the ***design*** button on the toolbar. Sizer automatically designs the member.
2. You will be asked to enter a file name for your project.

View Results

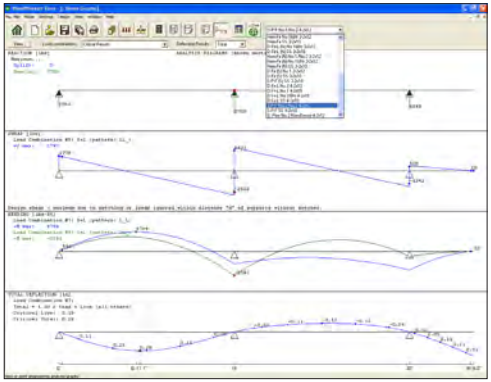
1. Click the up and down arrows to scroll through the Design Results output.
2. To print these results, choose the ***print*** button from the toolbar.

NOTE: The output shown is based on the U.S. National Design Specification®.



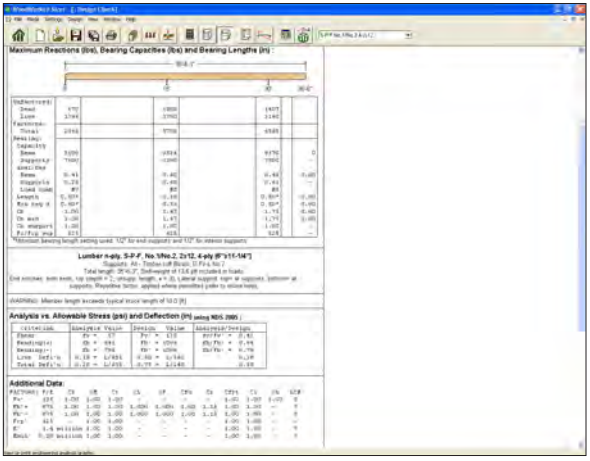
View Analysis Diagrams

1. Click the ***diagram*** button on the toolbar to view reactions, shear, bending moments and deflection diagrams.



Perform a Detailed Design on a Specific Section

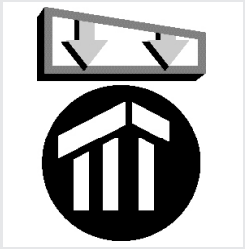
1. Choose S-P-F **No.1/No.2** 2-2X10 from the ***Suggested Beam Sections*** drop-down list on the toolbar.
2. Sizer will automatically perform a detailed design on the specific section chosen.



3. The results for the specified section are now displayed as shown.
4. You can repeat the above steps to perform a detailed design on any other glulam section listed in the ***Suggested Beam Sections*** drop-down list.

Designing a Column

A.3



3.1 Column Mode

What It Does

Column mode allows you to quickly enter and design individual wood columns, walls and beam-columns under multiple load conditions. Column mode provides extensive information about a single column or wall in your project. Typically, you would use Column mode to verify the design of critical members.

To select Column mode, click ***new*** on the toolbar and choose ***column***.

When Column mode is active, a checkmark is displayed next to Column on the Mode menu.

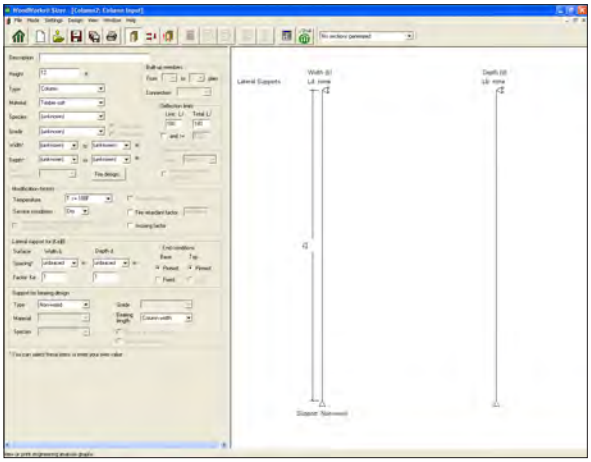
Transfer of a Column (or Wall) from Concept Mode
If you are operating in Concept mode and you wish to further analyze a particular column or wall, click the column or wall to select it and then choose ***Column*** from the ***Mode*** menu, or click the ***mode*** button on the toolbar.

Sizer first determines the axial design loads for the column or wall in question, based on the structure entered in Concept mode. Sizer automatically transfers the load information to the input fields of Column mode. For walls, it transfers the load information for a single wall stud.

Note: The transfer from Concept to Column mode is one-way. Any changes you make to the transferred member cannot be exported back to the original member in Concept mode. There are a few exceptions, such as the selection of **Self Weight**: "manually input " or "automatically included in loads analysis". Refer to **Help** on "*Apply options to Concept Mode*" which refers to what options can be tranferred with a click of a button in the ***Loads*** view, activated if Concept mode is open.

3.2 Column (or Wall) Parameters

Click ***column*** on the toolbar to describe the column (or wall) to be designed.



Height
This field specifies the height of the column or wall in the current units.

A height must be specified in order to continue with the design.

Type
This drop-down list specifies the type of compression member (Column or Wall) to be designed. The default selection is Column.

Description
A description of the column can be entered which will appear in the ***Design Results*** output and ***Design Check*** output.

Material
This drop-down list specifies the material database to use. Standard choices include timber post, glulam and a number of others. The default for columns is Timber-soft. The default for walls is **Lumber Stud**.

Species
This drop-down list specifies the species of wood to use to design the member. Available species depend on the selected database. For example, common choices for softwood timber are D.Fir-L, Hem-Fir, S-P-F, and S.Pine.

The default selection is *(unknown)* which forces Sizer to select suitable sections from each species.

Grade or Combination
This drop-down list specifies the grades of wood to use to design the member. Available grades depend on the selected database. For example, common choices for timber are No.2, No.1, and SS.

The default selection is *(unknown)* which forces Sizer to select suitable sections from each grade.

This list box is unavailable if the species is *(unknown)*.

Width and Depth
The two width drop-down lists specify the width range of the sections Sizer should use when designing the member. Similarly, the two depth drop-down lists specify the depth range. Available widths and depths depend on the selected database, species and grade.

The default selection for each of these drop-down lists is *(unknown)*, which forces Sizer to select from a full range of section sizes.

You can limit the section sizes to be considered by selecting appropriate width and depth ranges in the drop-down lists.

These drop-down lists are unavailable if the grade is *(unknown)*.

To select a custom section, enter some non-standard size (in actual dimensions) in each drop-down list. Note that a custom size may not be commercially available.

Note: Sizer performs a *design check* rather than a *design selection* if both the width range and depth range specify single values. Sizer cannot design a section if only the width or depth is a custom size. For example, you cannot have one field as *(unknown)* when the other contains a custom size.

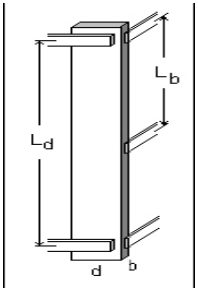
Built-up Members
If the selected material can be used in built-up sections, these two drop-down lists specify the range of plies to use. The default selection is *(unknown)*, which forces Sizer to select suitable sections of a range of plies.

For built-up members, the connection type used to assemble the member can also be specified. This affects the resistance of a member.

If the material cannot be used in built-up sections, these lists are disabled.

Deflection Limits
These fields allow you to specify the deflection limits to be used for design, based on the span (example L/360) and in absolute terms (1 inch). Default deflection limits are entered in **Settings [Design]** tab. See also the "**Loads**" button and "*Load types and combinations*" for long term deflection (creep) and load reductions (US only).

Lateral Support Spacing and End Connections
Select *Pinned* if the column base does not resist bending, or *Fixed* for a column base that does resist. For fixed column base, select pinned or free for the column top. Free column tops do not resist bending or translation.



K_eL for Width
The L_b field specifies the unsupported length associated with width b. The K_e field specifies the effective length factor. Its default value is 0.8 for columns with a fixed base, and 1.0 for columns with a pinned base. For free tops, the default value is 2.1 for U.S. and 2.0 for Canada.

K_eL for Depth
The L_d field specifies the unsupported length associated with depth d. The K_e field specifies the effective length factor. Its default value is 0.8 for columns with a fixed base, and 1.0 for columns with a pinned base. For free tops, the default value is 2.1.

Glulam Lay-up (U.S. Only)
This field allows the input of wane for certain combinations, with associated strength reduction. It also allows the input of non-edge bonding with associated strength reduction.

Fire Design Button (U.S. Only)
Clicking on this button opens the Fire Resistance and Treatment dialog that allows you to specify fire endurance design criteria and a fire-retardant
factor.

Fire Endurance Rating
Sizer checks the Fire Endurance Rating of timber and glulam beams or columns. Sizer calculates the *Fire Endurance Rating* based on the number of exposed sides and on the loads applied to a member. To activate the fire endurance check, the number of exposed sides must be defined. If the number of exposed sides is set to zero, the program will not perform the fire endurance check.

Fire Endurance Rating is activated through the **Settings/Design** tab.

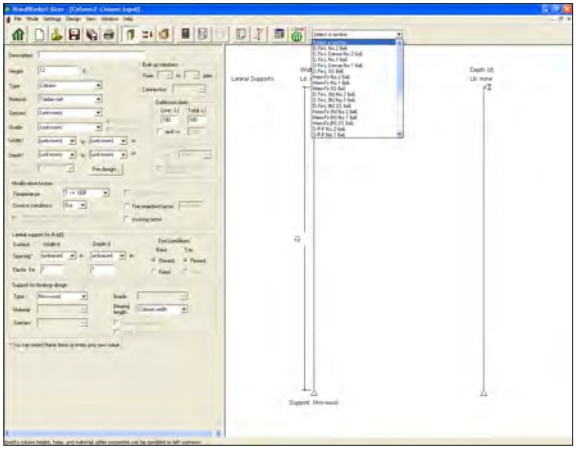
Service Condition
This drop-down list specifies either Wet or Dry service conditions for the column or wall. The default is **Dry**.

Temperature (U.S. Only)
The default is T<100°F. Other values for T are 100°F<T<125°F and 125°F<T<150°F.

Support for Bearing Design
These fields allow the user to choose the type, material, species, grade and bearing length for the column or wall stud. These fields cannot be left as unknown. The results will be shown in the ***Design Results*** to let the user know if the support bearing is sufficient for the column. For example, *a bottom plate* supporting a lumber stud can be designed if the *Type* is appropriately selected. If "*Same as wall stud*" is not selected, the bottom plate's specific *material*, *species*, and *grade* can be entered. otherwise the same materials as the wall stud is used.

The checkbox "*Bearing at Support End*" indicates that the supporting beam, sill plate, or wall bottom plate ends at the column or wall, so that the bearing length factor C_B is not applied.

Fire-Retardant Factor
This field permits you to specify the reduction in compression, bending, shear, and deflection resistance of wood treated with fire-retardant chemicals.



Treatment (Canada Only)
When you select Fire-retardant treatment or Preservative treatment from the drop-down, an input field becomes active for you to input the corresponding modification factor.

Stud Spacing
Active for walls only, this drop-down list specifies the stud spacing. You can select one of the three standard spacings of 12 in. (300mm), 16-in. (400mm), or 24 in. (600mm), or enter your own value.

Repetitive Member Factor (U.S.)/ Load Sharing (Canada)
This control allows you to specify if a sharing factor is to be applied.

Incising Factor (U.S. Only)
This field allows you to specify whether an incising adjustment factor should be applied.

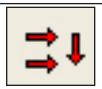
Selecting Sections for Design Check

Sizer provides you with an extensive list of suggested sections for the column it is designing. To get more details about a single section size, you can perform a design check on that section.

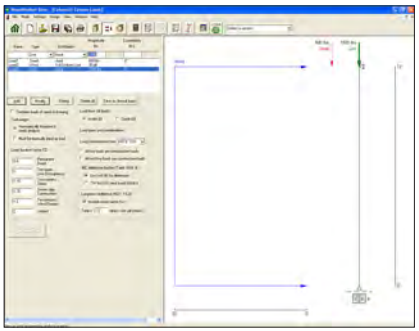
After running an initial design, select the section you want to check from the drop-down list entitled ***Suggested Column Sections*** at the right side of the toolbar. Sizer automatically fills in the Species, Grade, Width and Depth fields with the appropriate values. Sizer also automatically performs a design check on the selected section.

The list of suggested sections on the toolbar remains available until you perform another design (rather than a design check).

3.3 Loads



Click ***loads view*** on the toolbar to specify the loading of a column that is to be designed.



Name
Use the Name field to enter the name of the load you want to apply to the column. (Sizer generates an appropriate name if you leave this field blank.) The list box beneath this field contains the names of all the loads you have specified for this column. Click one to select it.

Type
This drop-down list specifies the type of load being applied. Sizer allows you to select from a variety of load types, including dead, live, roof live (U.S. only), snow, wind, impact (U.S. only), sustained live (storage and contained fluids) and dead (soil) (Canadian only), earthquake, and hydrostatic (Canada only, columns only). The U.S. version also allows all live loads or all roof live loads to be considered a "construction" load, and will use the appropriate duration factor if this is selected.

Depending on the load type specified, Sizer will apply the correct load duration factor and load combination factor to each load combination. For the Canadian version, Sizer also applies the correct load factor. The default load type is **Dead.**

Distribution
This drop-down list specifies the type of load distribution: Axial, Full Uniform Line, Partial Line, Full Uniform Area, Partial Area, Triangular, Trapezoidal, Point and Applied Moment.

Axial loads are applied to the top of the column, where a positive value loads the column in compression. The remaining loads are applied laterally and load the column as a beam.

For walls, an axial load is entered as a uniformly distributed line load along the top of the wall.

Sizer applies lateral loads to the face identified in the Load Face box of the Column screen.

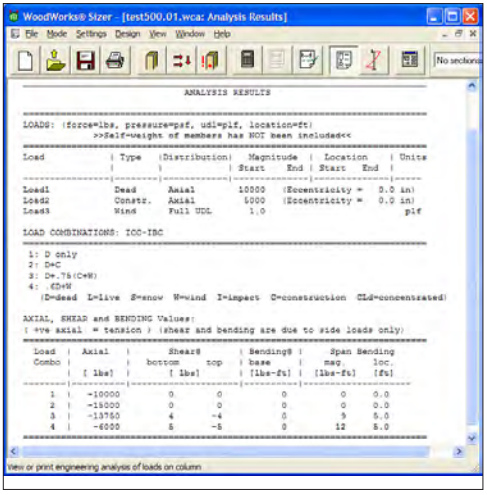
Magnitude
This field specifies the magnitude(s) of the column load at the beginning and end of the loaded portion of the member (For a Point load, only one load magnitude is specified.). True Uniform loads should have the same magnitude at point a and b. Trapezoidal loads usually have different values for the two magnitudes.

Note that area loads are converted to and displayed as line loads (plf or kN/m) on the load diagram.

Eccentricity
This field applies only to axial loads, with the same eccentricity for all. The bending effect is the same for an eccentricity and a lateral load, when both have positive values.

Location From Bottom
This field specifies the location of the selected load. The information required depends on the type of load distribution.

For lateral loads, distance is measured from the bottom of the column. For axial loads, eccentricity is measured from the center of the column.



Add Loads of Same Type
This option only applies to the loading diagram. When enabled, individual loads are accumulated into a single loading profile for each load type (dead, live, etc). When not enabled, loads overlap with other loads of the same type. The selected load appears in bold.

Loads and Load Combinations

Load Types and Combinations
Sizer combines loads based on the Allowable Stress Design method using the basic load combinations in the IBC/ASCE (US) and Limits States Design load combinations (Canada). In addition to considering the building code load combinations based on the load types present, Sizer also creates load combinations to consider the affects of pattern loading live type loads for multi-span beams or joists when pattern loading is active.

Load Duration Factors (U.S. Only)
Sizer applies load duration factors according to the load type. These factors are set to the standard NDS (U.S. version) or CSA O86 (Canadian version) values by default but can be changed using this form. (Note that in the Canadian version, when D > L, Sizer determines the default factor for standard term loads according to the equation shown in CSA O86-01 4.3.2.3 - refer to the online help for details).

Apply Options to Concept Mode
This button in Beam Load View and Column Load View, is only active when there is a Concept mode file open, and allows the application of the following settings to have an affect on Concept mode. This items can only be accessed from Beam or Column mode, and are:

- Self-weight
- Line Loads applied over design span only
- Long-term deflection factor
- Use L+0.5D for deflection
- Load duration factors CD
- Load combinations from…
- Temperature (from beam view)

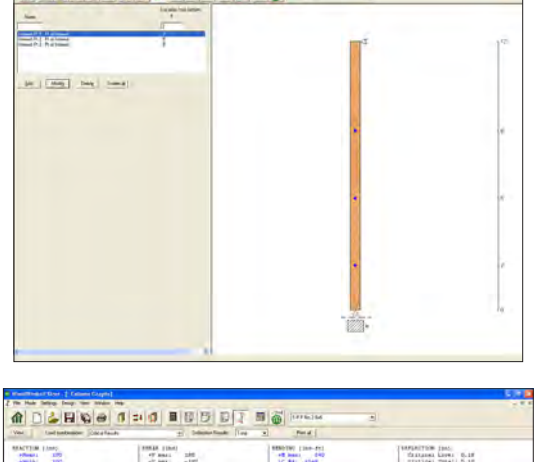
Self-Weight
If *"automatically included in loads analysis*" is selected by the user, the self-weight of a member is considered in the design. The actual effects of self-weight are calculated by multiplying the specific gravity of the material times the area of the section. This load is then added to the dead load when checked. Alternatively the self weight can be ignored or manually added by selecting "*Must be manually input as load*".

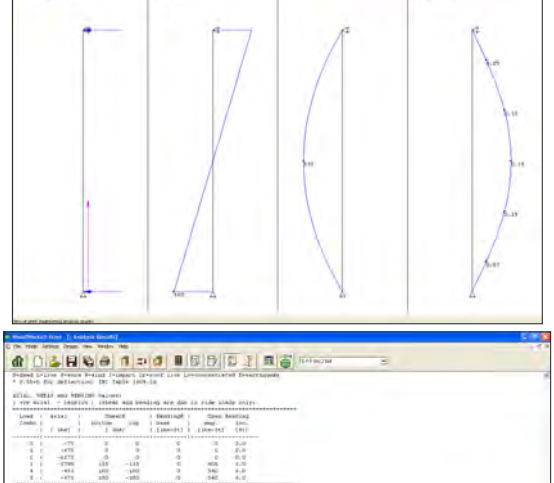
Load Face
Select ***Width*** if lateral loads such as wind are applied to the narrow face of the member, or ***Depth*** if applied to the wide face. All lateral loads and eccentric axial loads cause bending about the same axis.

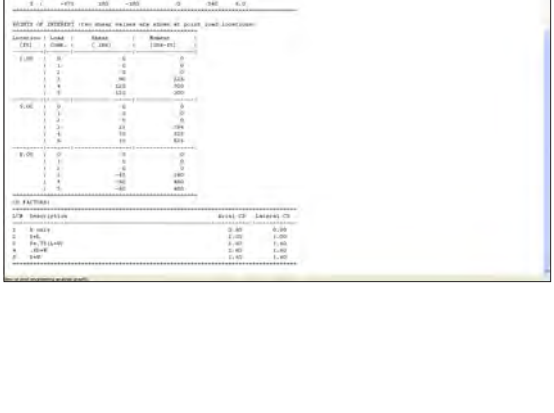
Deflection Factors (U.S. Only)
There are two leniencies offered in the IBC related to deflection. These optional IBC deflection factors can be selected using two checkboxes. The first allows the deflection resulting from L + D to be calculated using L+ 0.5D for wood structural members having a moisture content under 16 percent at the time of installation and used in dry conditions. The second option is related to components and cladding wind loads, where deflection may be calculated using only 70 percent of the wind load.

The NDS provides guidance on considering the effects of long term deflection by increasing the contribution of dead load by a creep factor, Kcr, that typically ranges from 1.5 to 2; any value can be entered if the "*Include creep factor*" is checked.

3.4 Points of Interest







Click the ***points of interest*** button on the toolbar to investigate the shear and moment at any point along the length of column or beam. A point of interest is generated by specifying a location to perform the analysis. Now click ***Add*** to add this to the list. Several points of interest can be specified.

After performing a design, the point of interest result will be shown in the Diagrams window and in the Analysis results output.

3.5 The Design Process

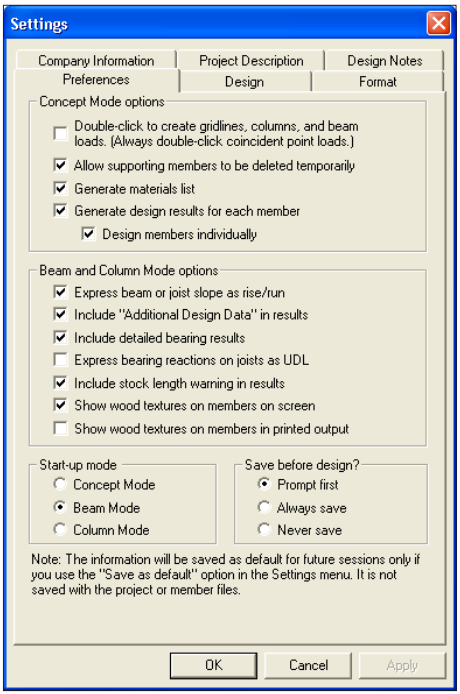
Starting the Design Process

To start the design process, click ***design*** on the toolbar. Sizer performs an analysis and design of your column with the information you entered and automatically displays the results.

If you specified *(unknown)* for some entries, Sizer selects a series of suitable sections that provide acceptable results. If there are no *(unknowns)* and you have not specified ranges in the width or depth fields, Sizer does a design check on the specified section and verifies that the results are within the selected design code's limits.

The number of sections that Sizer selects depends on how many fields you specified as *(unknown)* in the column screen.

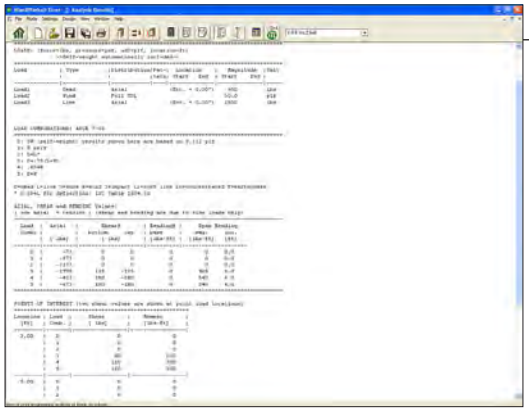
Sizer prompts you to save the current project prior to doing the design. To change this, choose one of the ***Save Before Design*** options from the ***Settings*** menu.



Save Before Design (Prompt first) Tells Sizer to prompt you to save the current worksheet prior to starting the design process. This is the default.

Save Before Design (Always save) Tells Sizer always to save the current project automatically (without prompting you) prior to starting the design process.

Save Before Design (Never save) Tells Sizer not to save the current project prior to starting the design process. Sizer will not even prompt you to save.

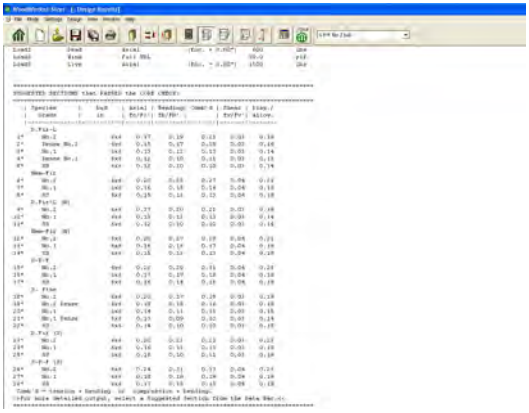


Viewing and Printing the Results

Once Sizer has designed the column, it creates results files. Sizer then gives you several options for viewing the results.

Analysis Results

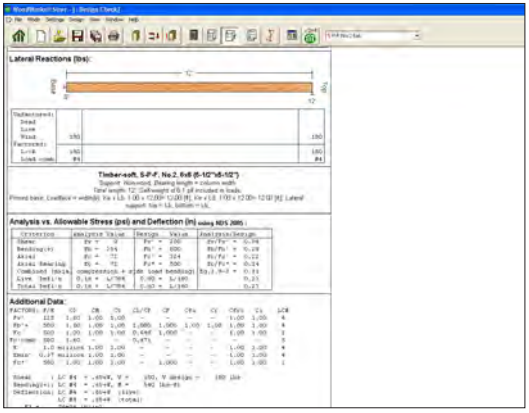
Click the ***analysis*** button on the toolbar to see the analysis results (maximum shears, bending moments and so on) for your column, beam-column or stud wall in tabular form.



Design Results

If you specified any field as *(unknown)* for your column or stud wall, Sizer computes all the possible column, beam-column or stud wall sections during the design process. It also computes the ratios of analysis/design values for axial, bending, combined axial and bending, shear, and displacement for each section.

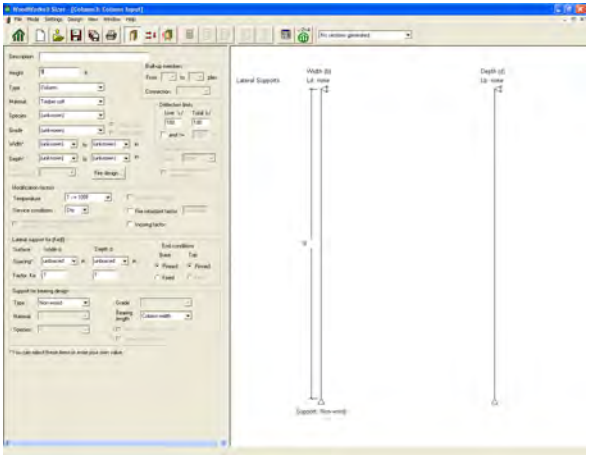
To see these results, click on the ***results*** button on the toolbar menu or choose ***Design Results (Suggested Sections)*** from the ***View*** menu.



Design Check

If you requested a specific column, beam-column, or stud size, Sizer performs a design check and computes analysis and design values (for example axial, shear, and bending moment) for the section.

To see these results, none of the parameters can be left as *(unknown)*. Then click ***check*** on the toolbar.



Printing Results

To print results, click on the ***print*** button when the result file or diagram screen that you wish to print is on screen. To batch print the analysis diagrams for all load combinations, press ***Print All.***

Analysis Diagrams

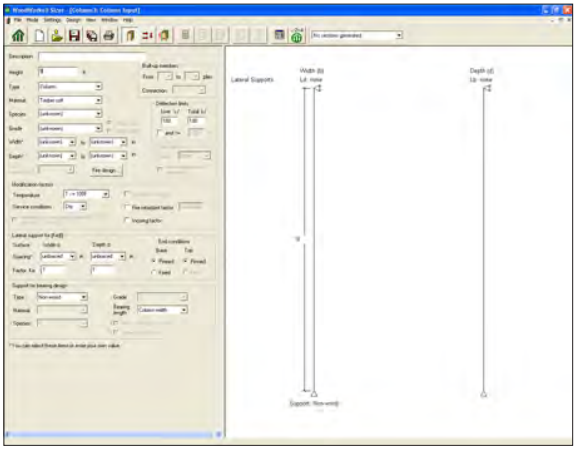
Click the ***diagram*** button on the toolbar to view the analysis diagrams.

Sizer creates five analysis diagrams; the load Envelope, Support, Shear, Bending, and Deflection diagrams.

To print the current diagram, click the ***print*** button on the toolbar

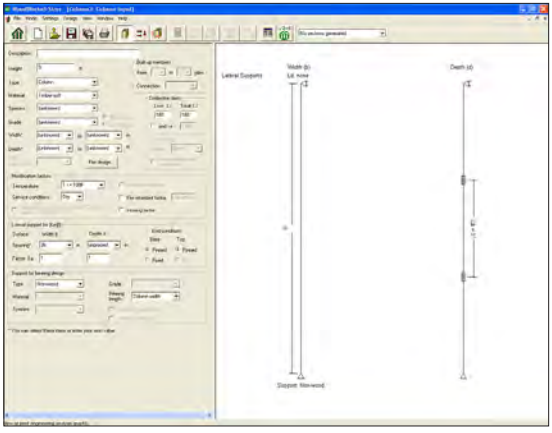
- ***View…*** – This button controls which analysis diagrams are shown, including the Load Envelope, Reaction, Shear, Moment and Deflection diagrams. The Load Envelope diagram is not available when *Critical Results* is selected as the load combination.
- ***Load Combinations*** – The diagram view results shown are based on the load combination selected in this pull-down menu. When Critical Results is selected, the analysis diagrams shown are for the worst case results of all load combinations. Load combination numbers shown correspond to those used throughout Sizer (Analysis Results, Design Check, diagrams, etc.).

3.6 Tutorial

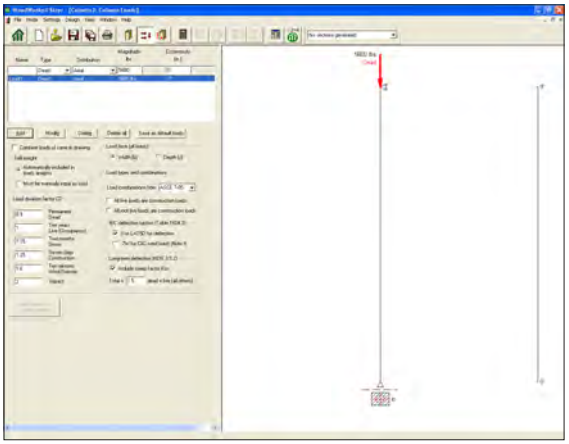


Defining the Parameters

1. Start the program in Column mode.
2. Select the ***Height*** field and enter a height of **9** (ft).

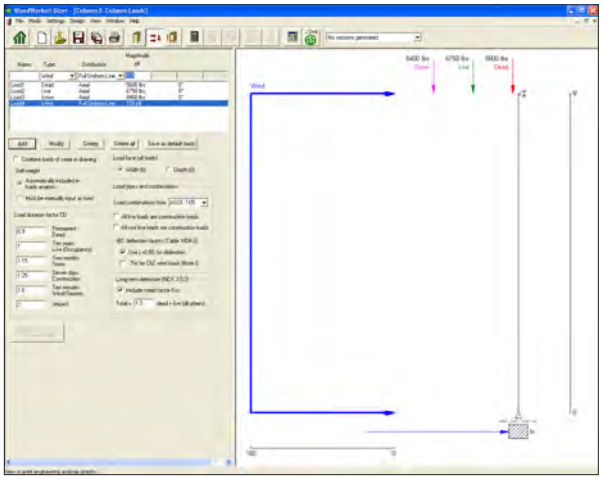


3. Under Lateral Support Spacing for K_eL, select ***Width*** (L_b) field and enter **36** (in) for the unbraced length in the narrow direction.

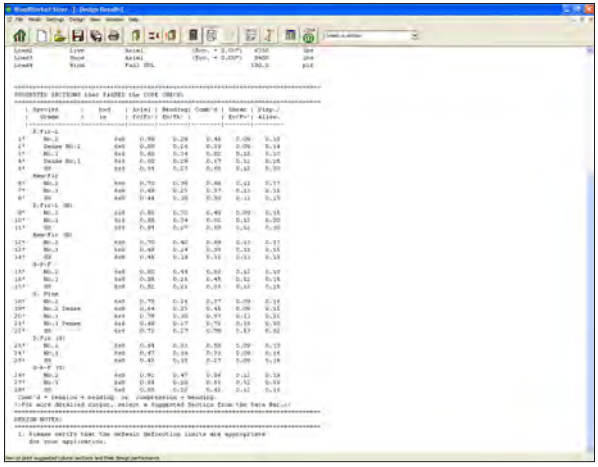


Loading the Column

1. Click ***load*** on the toolbar.
2. Choose ***Dead*** from the ***Type*** drop-down list.
3. Specify ***Axial*** as the load ***Distribution***.
4. Select the ***Magnitude*** field and enter a magnitude of 5600 (lbs).
5. Click ***Add***.



6. Repeat steps 2 to 5 for the following:
   Snow, 8400 (lbs)
   Live, 6750 (lbs)
7. Choose ***Full Uniform Line*** from the ***Distribution*** pull-down.
8. Select the ***Magnitude*** field and enter a magnitude of **150** (plf).
9. Choose ***Wind*** from the ***Type*** drop-down list.
10. Click ***Add***.



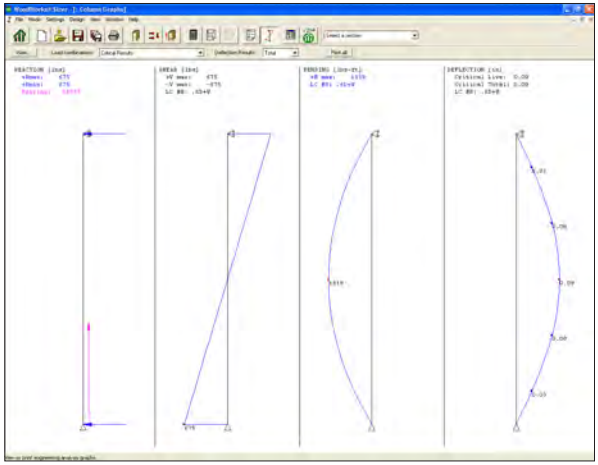
Designing the Column

1. Click the ***design*** button on the toolbar. Sizer automatically designs the member.
2. You will be asked to enter a file name for your project.

View Results

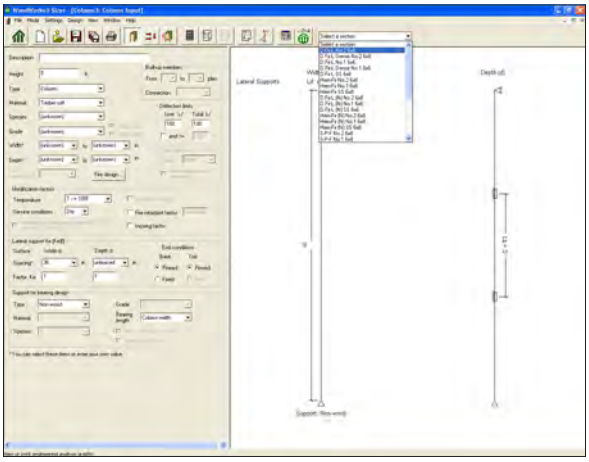
1. Click the up and down arrows to scroll through the results.
2. To print these results, click the ***print*** button on the toolbar.

NOTE: The output shown is based on the most recent U.S. National Design Specification® for Wood Construction.



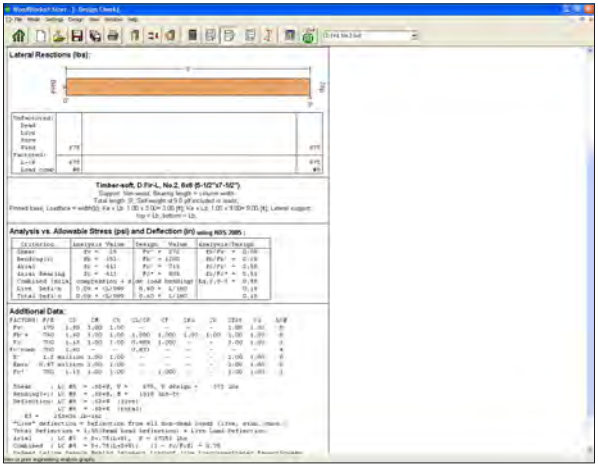
View Analysis Diagrams

1. Click ***diagram*** in the toolbar to view reactions, shear, bending moments and deflection diagrams.



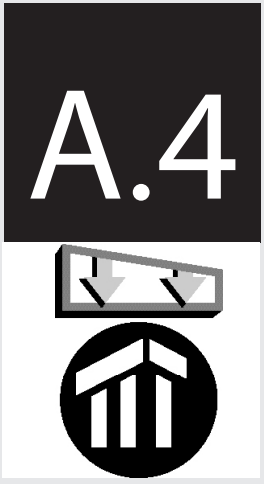
Perform a Detailed Design on a Specific Section

1. Choose ***D.Fir L > No.2> 6-x 8*** from the ***Suggested Sections*** drop-down list on the status bar. Sizer automatically performs a detailed design for this section.



2. Click the up and down arrows to scroll through the results.
3. Repeat step 1 to perform a detailed design on any other section listed in the ***Suggested Sections*** drop-down list.

Designing a Structure



4.1 Concept Mode

By default, Sizer starts in Concept mode. If you wish, you can reconfigure Sizer to start in Beam mode or Column mode by changing the default ***Startup Mode*** option in the ***Settings*** menu under ***Preferences***. To switch to Concept mode at any time, choose ***Concept*** from the ***Mode*** menu.

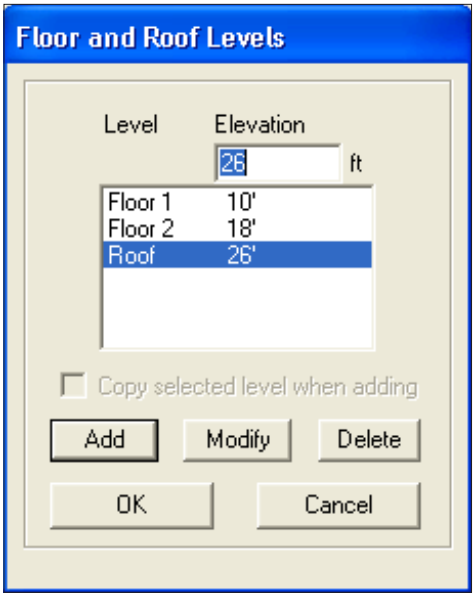
The Concept mode graphical work area allows you to quickly design a complete structure using columns, walls, beams and joists. The Concept mode will only design for gravity type loads. These loads must be applied as uniformly distributed line or area loads.

The Concept mode is meant to be used as a preliminary design step. For a more detailed design, individual members can be imported from the Concept mode to the Beam or Column modes.

Click items in the work area to select them. If you hold down the CTRL key when you click, you can add or remove items from the currently selected set. You can choose ***Select All*** from the ***Edit*** menu to select all of the items in the current view.

4.2 Levels

Click on the ***level*** button on the toolbar menu. The Levels dialog opens and allows you to specify the elevation for each level of your structure.



To add level elevations, enter a new value and click ***Add***. The maximum number of new levels is six.

When design of a structure has already begun, the option of copying the selected levels to the new ones is visible. Sizer will not, however, convert roof joists to floor joists or vice versa.

To modify any existing level elevation, select it, enter a new value and click ***Modify***.

To delete a level elevation, select it and click ***Delete***.

Click ***OK*** to save your changes or click ***Cancel*** to discard your changes.

The Levels dialog is also used to change to different levels. Select a level by clicking on it so that it is highlighted. Click ***OK*** and Sizer will change to the selected level. Another way to change to different levels is by using the drop-list in the toolbar.

4.3 Design Groups

Why they are used

The Sizer design process is based on “design groups”. Design groups allow you to group similar members together for the design process. Sizer determines the lightest section size that can be used for each of the design groups, based on the most critical member in each group.

You can make your group divisions as coarse or as fine as you wish. Anything from a single group for all beams to a separate beam group for each beam is acceptable. However, a single group for all beams will likely result in excessively large section size for many beams. Similarly, a separate beam group for each beam (while determining the optimal size for each beam) results in an excessively large number of different sections for construction.

How to create them

Design groups are created using a dialog, as described in the following sections. To help you get started, Sizer provides default design groups for each major structural element:

- Column for columns
- Wall for walls
- Beam for beams
- FloorJoist for floor joists
- RoofJoist for roof joists

Group Dialog

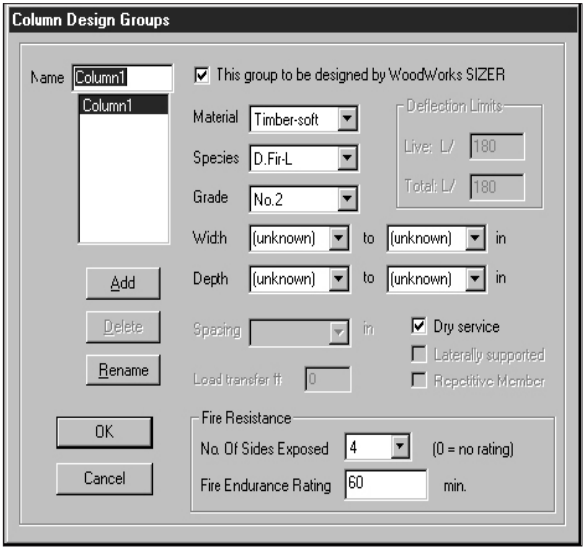
To create a new group, enter appropriate values in the dialog fields and then click ***Add***.

To change a design group, select the group from the list, change any of the values in the dialog fields and then click ***Modify***.

To delete a design group, select the group from the list and then click ***Delete***.

Column Groups

Click ***column*** on the toolbar to change the view to Columns. Now click ***Groups*** on the toolbar. The dialog box named *Column Design Groups* opens.



Column Dialog Items

Name
This field contains the name of the currently selected group. The list beneath contains all the column groups created. Click on a group to select it. Once highlighted, you can modify or delete it.

To Be Designed
Select this check box if the design group is to-be-designed. Leave it unchecked if the design group is not-to-be-designed. Unchecking it disables all field selections. Either design group supports and transfers loads.

Dry Service
This check box is checked if the service condition of the design group is dry. If the service condition is wet, this box should be unchecked.

Material
This drop-down list specifies the material database. The standard choices include timber, multi-ply lumber, glulam and a number of others. The default for columns is **Softwood Timber.**

Species
This drop-down list specifies the species of wood to use for the design process. Available species depend on the selected database, but common choices for timber are D.Fir-L, Hem-Fir, S–P–F and S.Pine. The default is the first species in the list.

Grade
This drop-down list specifies the grades of wood to use for the design process. Available grades depend on the selected database, but common choices for timber are No. 2, Dense No. 2 (U.S. Only), No.-1 and Dense No.1 (U.S. Only). The default is No. 2.

Width
These two drop-down lists determine the width range of the sections to be considered in the design process. Available widths depend on the selected database, species and grade. The default selection is *(unknown)* for each of the lists, which forces the design process to select from the full range of section widths. To specify a limited range of widths to be considered, select a different width in either or both of the lists.

Depth
These two drop-down lists determine the depth range of the sections to be considered in the design process. Available depths depend on the selected database, species, grade and width. The default selection is *(unknown)* for each of the lists, which forces the design process to select from the full range of section depths. To specify a limited range of depths to be considered, select a different depth in either or both of the lists.

No. Of Sides Exposed (U.S. Only)
Select the number of sides of the member that are exposed for calculating fire endurance rating. The selections are 0, 3 or 4 sides. Select 0 to indicate that the member is fully sheathed and no fire resistance calculation is to be performed.

Fire Endurance Rating (U.S. Only)
Enter the required fire endurance time. Do not enter a value greater than 60 minutes as this falls outside the research upon which the methodology is based. Sizer will attempt to find a section whose fire endurance meets or exceeds the value you enter, according to IBC 721.6.3.

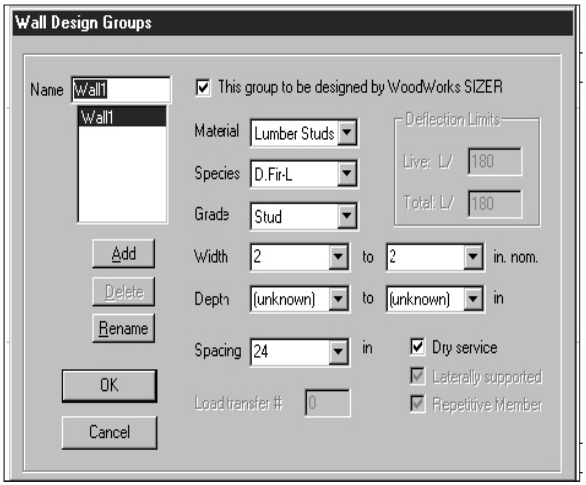
Column Assumptions

Note that Sizer defaults to the following assumptions:

- Columns are pinned at both ends.
- Columns are laterally restrained only at the ends.

Wall Groups

Click ***wall*** on the toolbar to change to Walls view. Now click ***Groups*** on the toolbar. The dialog box named *Wall Design Groups* opens.



Wall Dialog Items

When creating design groups for walls, you use a dialog similar to the one used to create design groups for columns. The additional information in the beam dialog is described below.

Spacing
This drop-down list specifies the stud spacing. You can select one of three standard spacings or enter your own value.

Repetitive Member Factor (U.S.)
Load Sharing System Factor (Canada)
Sizer assumes that all walls act as a system and as a default applies the repetitive member factor C_r for the U.S. or the system factor K_H for Canada.

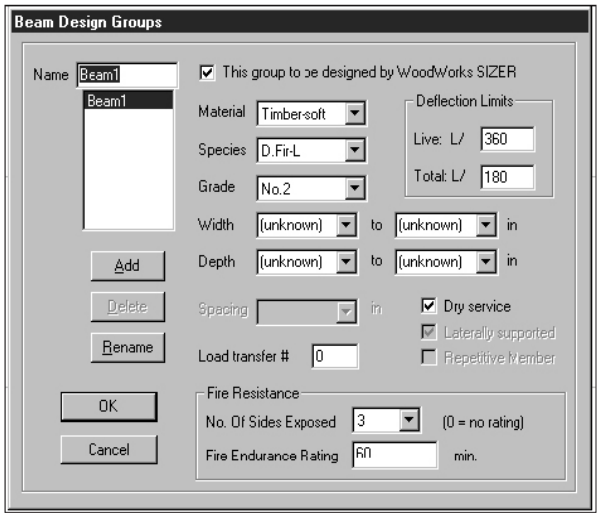
Wall Assumptions

Note that Sizer defaults to the following assumptions:

- Walls are pinned at both ends.
- Walls are prevented from buckling about the weak axis by wall sheathing.

Beam Groups

Click ***beam*** on the toolbar to change to Beams view. Now click ***Groups*** on the toolbar. The dialog entitled *Beam Design Groups* opens.



Beam Dialog Items

When creating design groups for beams, you use a dialog similar to the one used to create design groups for columns and walls. The additional information in the beam dialog is described below.

Material

In addition to the standard choices such as timber and glulam, new material databases have been added to include SCL (Structural Composite Lumber) materials for beams only. This material is more commonly known as Parallel Strand Lumber (PSL), Laminated Strand Lumber (LSL) or Laminated Veneer Lumber (LVL).

Deflection Limits

The Live field specifies the maximum allowed deflection of the design group member for live loads.

The *Total* field specifies the maximum allowed deflection of the design group member for total loads.

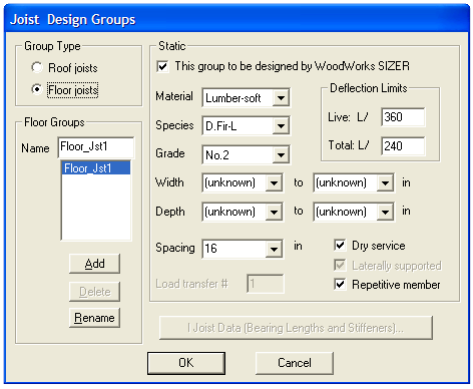
Laterally Supported

Select this check box if the top of the beam is laterally braced continuously while the bottom is laterally restrained only at supports.

Note: This selection is only available for glulam (Canadian only). For all other materials, notes are provided in the output specifying the necessary lateral support.

Load Transfer Number (beam-to-beam)

This field determines how loads are transferred from beams of one group to beams of another group. Loads always transfer from a group with a higher load transfer number to a group with a lower load transfer number. For instance, in order to transfer the load from one beam directly to another beam the supported beam needs a higher Load Transfer number than the supporting beam. Use numbers between 0 (default) and 99. The actual Load Transfer number is not significant, a supported member only needs to have a higher number than the supporting member.



Joist Groups

Click ***joist*** on the toolbar to change the view to Joists. Now click ***Groups*** on the toolbar. The dialog entitled *Joist Design Groups* opens.

Floor Joist Vibration (Canada Only)

When using floor joists, the sheathing thickness, lateral support and connection to subfloor can be selected from a drop-down list for the determination of floor vibration

Joist Dialog Items

When creating design groups for joists, you use a dialog similar to the one used to create design groups for columns, walls and beams. The additional information in the joist dialog is described below.

Material

Standard choices for joists include I–joists and dimension lumber. Visually graded dimension lumber is referred to as Lumber and machine stress rated dimension lumber as MSR Lumber. The default is Softwood Lumber.

Grade

Available grades depend on the selected database, but common choices for lumber are Select Structural, No.-1 and No. 2. The default is No. 2.

Spacing

This drop-down list specifies the joist spacing. Select one of three standard spacings or enter your own value.

Design Groups Summary

Once you have specified your design groups, you can create your structure in the Concept mode.

4.4 Gridlines and Gridpoints

Gridlines are horizontal (East-West) or vertical (North-South) lines positioned on the work area. Gridlines appear as dark blue dashed lines in the Grid view. The intersection of a North-South gridline with an East-West gridline is a *gridpoint*.

When you change from Grid view to any other view, the gridlines appear as light gray dashed lines.

Under ***View*** in the ***Settings*** menu, a snap increment option is available for gridlines. A snap increment allows you to determine the smallest increment a newly created gridpoint will move or "snap" to.

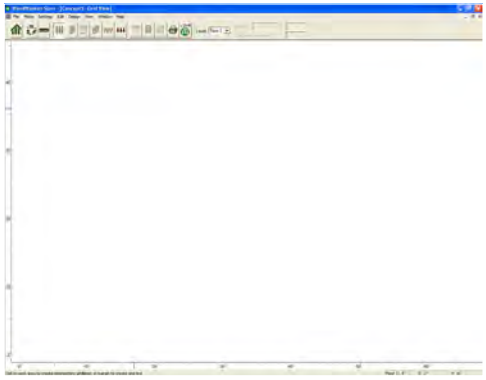
Automatic Generation

This feature makes a simple task of producing a screenfull of gridlines at regular intervals. First start a new **Concept** mode project. Next, from the menu bar select **Settings|Change...|View** and enter the desired gridline interval as the N-S and E-W snap increments. Press the **Ok** button and from the menu bar select **Edit|Generate Grid.**

Manual Generation

Click ***grid*** on the toolbar. The ***grid*** button on the toolbar appears as light gray to indicate the view. The Sizer window title also indicates the current view.

Point close to where you want to place a gridpoint and click to create it.



The first time you click in the work area, Sizer creates a North-South gridline designated **A** and an East-West gridline designated **1**. The intersection of these gridlines is at gridpoint **A.1.** (Gridline intersections are referred to as **character.number**.)

If you specified a snap increment, the gridpoint snaps to the nearest increment.

To create more gridpoints along an existing gridline, click at the required position along the existing gridline. You can place as many gridpoints as you wish in the work area.

Moving Gridlines
To move a gridline, click on it and hold the mouse button and drag it (which changes to red) to the position you want. Use CTRL-click to select multiple gridlines. Dragging any one of the selected gridlines moves all of them.

To deselect a gridline, CTRL-click it. To deselect all selected gridlines, click any blank spot in the work area.

Removing Unwanted Gridlines
To delete individual gridlines, select them by clicking on them so that they appear in red and then press the DELETE key on the keyboard.

You cannot delete a gridline that has a wall or column located on it.

Should you accidentally generate a screenfull of gridlines you can easily delete them. First make sure that the **Grid** button on the toolbar is the active button. From the menu bar select **Edit**|**Select All**. Then press the Del key. Sizer will delete all unused gridlines and will issue a warning for each gridline that is in use and cannot be removed.

Selecting the Generate Gridlines item on the ***Edit*** menu automatically generates gridlines over the whole viewing area. The spacing between gridlines equals the current snap increment, which is set under the **View** tab of the **Settings** options. If you make a mistake and generate gridlines at the wrong increment you need only use the ***Select All*** item of the ***Edit*** menu and then press the delete key. This will remove all gridlines that are currently not “in use” by members.

Concept Mode Data Bar

The Concept Mode Data toolbar is active in the grid view when a gridline or gridpoint is selected. The Concept Mode Data toolbar will display information on the selected item, including gridline name, gridline position and gridpoint elevation.

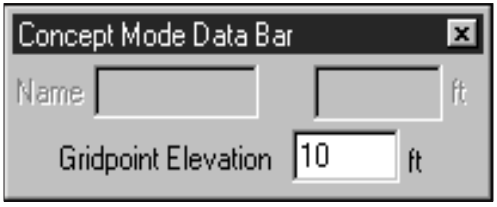
Gridline Name
If a gridline is selected, the Concept Mode Data toolbar will indicate the name of the gridline.

The gridline name can be changed by typing over the existing name included in the toolbar

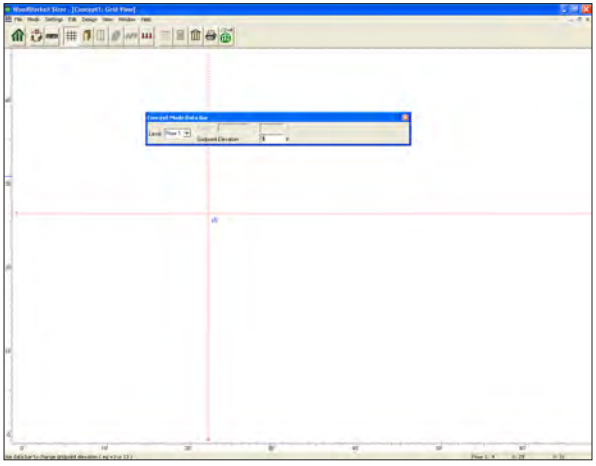
Gridline Position
If a gridline is selected, the Concept Mode Data toolbar will indicate the position of the gridline as the distance from the parallel axis (either X or Y). For example, if a gridline runs East-West, the position indicated will be the Y-coordinate.

The gridline position can be changed by typing over the existing name included in the toolbar.

Gridpoint Elevation
If a gridpoint is selected, the Concept Mode Data toolbar will indicate the gridpoint elevation.



The gridpoint elevation can be changed by typing over the existing elevation in the toolbar. If a gridpoint elevation is changed, a plus or minus elevation will appear beside the gridpoint on the **Grid** view to indicate its elevation in reference to the elevation of the current level.



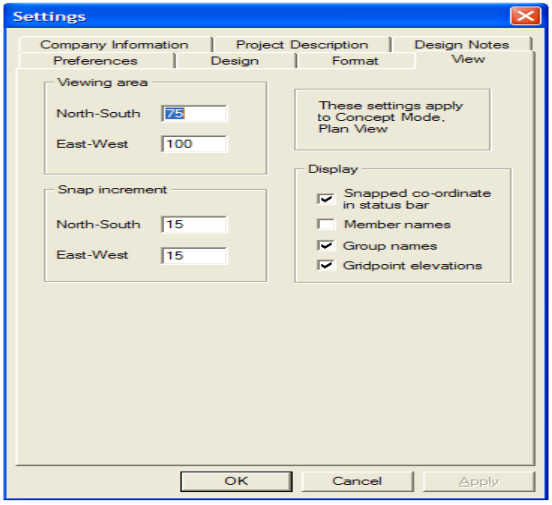
To create a structure with sloped members (such as a gable roof), you need to specify non-standard elevations for some members. In Sizer you specify gridpoint elevations from ground level.

Changing the elevation at a gridpoint changes a column's or wall's height at that point. Naturally, this affects the results of the design process.

The elevation of a gridpoint specifies the elevation of all beams and joists supported by the column or wall located at the gridpoint. You cannot have two beams at different elevations supported at the same gridpoint.

View Options

Choose the **View** tab from the Settings Dialog. This tab allows you to specify different viewing options such as the size of the viewing area, the snap increment, the percentage zoom, and member names.



Viewing Area

These fields specify the maximum viewing area in plan for the North-South and East-West directions.

Snap Increment

This specifies the smallest increment a newly created gridpoint will move or “snap” to. For example, a snap increment of 2.0 ft allows you to place gridpoints at 10.0, 12.0 and 14.0 ft, but not at 9.0 or 11.0 ft.

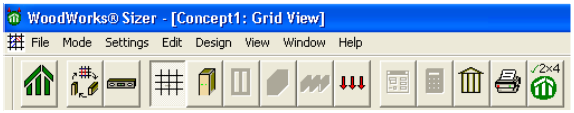
Display

These options allow you to include additional information on the diagrams for viewing or printing. This includes the snapped coordinates, the member names, and the group names.

4.5 Columns

Creating Columns

Click ***column*** on the toolbar. The ***column*** button on the toolbar appears as light gray to indicate the view. The Sizer window title also indicates the current view.



All to-be-designed columns appear as hollow blue squares, while not-to-be-designed columns appear as solid blue squares.

In Walls, Beams, Joists, and Loads view, columns appear as light grey squares.

Columns

New columns must be added to existing column groups. If you have created column design groups, simply choose the appropriate group from the drop-down list included in the Concept Mode Data toolbar. If you have not created the column design groups, click the ***groups*** button on the toolbar to access the dialog for creating column and wall groups. See section 4.3 for more details.

Columns must be created on a gridpoint. Click a gridpoint to create a column there. Click on any position other than a gridpoint is ignored.

Deleting Columns

Click a column to select it. Selected columns appear red.

To delete a column, select it and press the DELETE key on the keyboard.

Concept Mode Data Bar

The Concept Mode Data toolbar is active in the Column view when a column is selected. The Concept Mode Data toolbar will display information on the selected item, including the column name and the column group name.

Column Name
If a column is selected, the Concept Mode Data toolbar will display the column name.

As a default, the column name is **c1** for the first column, **c2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar

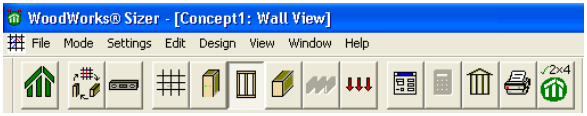
Column Group
If a column is selected, the Concept Mode Data toolbar indicates which column design group it belongs to.

The group of a column can be changed by selecting a different group from the drop-down list included in the toolbar.

4.6 Walls

Creating Walls

Click ***wall*** on the toolbar. The ***wall*** button on the toolbar appears as light gray to indicate the view. The Sizer window title also indicates the current view.



All to-be-designed walls appear as hollow blue lines, while not-to-be designed walls appear as solid blue lines.

In Columns, Beams, Joists, and Loads view, walls appear solid gray lines. In Grid view, the walls are shown as light grey lines.

Walls
New walls must be added to existing wall groups. If you have created the wall design groups, simply choose the appropriate group from the drop-down list at the right of the status bar. If you have not created the wall design groups, click ***groups*** on the toolbar to access the dialog for creating column and wall groups. See section 4.3 for more details.

To create a wall, drag from the start gridpoint to the end gridpoint. As you drag, Sizer displays a solid “rubber band” line showing where the wall will go. Release the mouse button at the gridpoint where you wish to end the wall. Sizer draws the wall between the two gridpoints.

You can create walls between any two gridpoints. One wall can cross (or intersect, if you prefer) another wall, but two walls cannot overlap. For example, you cannot create two walls between the same pair of gridpoints. Sizer indicates an error if you try to overlap walls.

Even if a wall spans several gridpoints, it is treated as continuous. However, if you create several walls in a row, spanning a series of gridpoints along one gridline, the walls are treated as discontinuous. This does not affect the design results unless the discontinuous wall segments belong to different wall design groups.

Previous versions of Sizer required that a wall or beam on a lower-story support an upper story wall. This limitation has been removed and joist areas can now support upper story walls in two ways:

End of a Cantilevered Joist Area
Walls can sit on the end of cantilevered joist areas. Please refer to Sizer’s online help and the examples in this document for further details.

Across the Interior of a Joist Area
A joist area where the joist direction is not parallel to a wall can now support that wall. The wall produces a uniformly distributed load which results in a point load on each joist in the joist area. Please refer to Sizer’s online help and the examples in this document for further details.

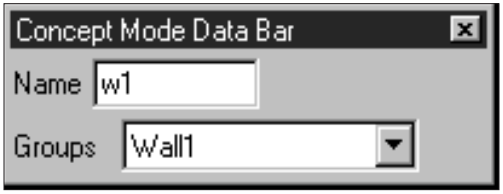
Deleting Walls

Click a wall to select it. Selected walls appear red.

To delete a wall, select it and press the DELETE key on the keyboard.

Concept Mode Data Bar

The Concept Mode Data toolbar is active in the Wall view when a wall is selected. The Concept Mode Data toolbar will display information on the selected item, including the wall name and the wall group name.



Wall Name
If a wall is selected, the Concept Mode Data toolbar will display the wall name.

As a default, the wall name is **w1** for the first wall, **w2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar.

Wall Group
If a wall is selected, the Concept Mode Data toolbar indicates which wall design group it belongs to.

The group of a wall can be changed by selecting a different group from the drop-down list included in the toolbar.

Examples of Walls Supported by Joist Areas
Basic Rules are:

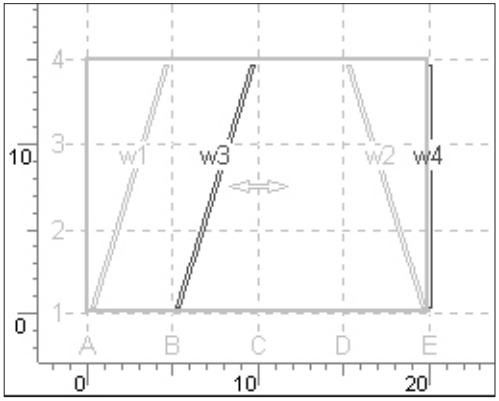
1. A Wall cannot be supported by a 3-sided joist area.

2. A wall can be supported by a 4-sided joist area on the floor below if the wall extends exactly from one edge joist to the other. This ensures that it is fully supported and that it loads all of the joists.

3. If the wall spans the interior of the joist area, it must be parallel to all but possibly one of the underlying supports of the joist area. Thus if there are four supports under a multi-span joist area, the wall must be parallel to at least three of them. This ensures that the outermost (or edge) joists in the joist area are the critical ones.

4. Walls can always be supported at the end of cantilevered joists.

Refer to the online help for more information about Wall Supports.



In this example there are two walls (w1 and w2) supporting a joist area on the first level. It is not possible to place a wall along gridline C because it would not be parallel to either w1 or w2 below. A wall on the second level must be parallel to at least 1 (i.e. 2 – 1 = 1) of the joist supports on the first level. This is the case for wall w3.

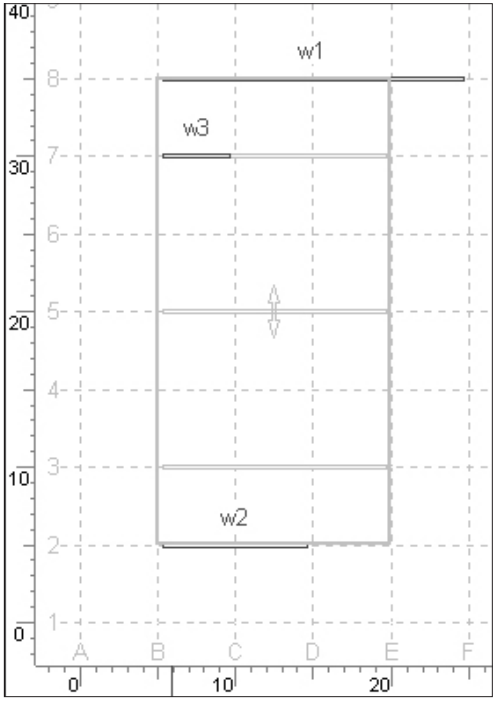
In the same example, a wall can be placed at the end of the cantilevered joist area, and such is the case for wall w4.

It is worth noting that, as in previous versions of Sizer, a second level wall could be placed directly over any of the first level walls. The load would then be transferred directly to the lower wall.

Partial Wall on joist area

Sizer allows partial walls to be placed on joist areas if there is a supporting wall below it or if it is at the end of a cantilevered joist area. However, these partial walls will not be carried by the joists, but extend to a wall or the ground below.

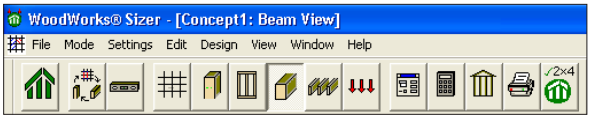
In this example, we are on the second floor. Wall w1 and wall w2 are allowed, but the joist area does not support them. They extend down to the ground. Wall w3 is supported by the wall below. An attempt to create a wall from B6 to C6 fails because it does not span the whole extent of the joist area. An attempt to create a wall from B6 to F6 fails, because part of the wall is not supported by the joist area.



4.7 Beams

Creating Beams

Click ***beam*** on the toolbar. The ***beam*** button on the toolbar appears as light grey to indicate the Beam view. The Sizer window title bar also indicates the current view.



All to-be-designed beams appear as hollow blue lines, while not-to-be designed beams appear as solid blue lines.

In Columns, Walls, Joists, and Loads view, beams appear as solid grey lines. In Grid view, beams appear as light grey solid lines.

Beams

New beams must be added to existing beam groups. If you have created the beam design groups, simply choose the appropriate group from the drop-down list included in the Concept Mode Data toolbar.

To create a beam, drag from the start gridpoint to the end gridpoint. As you drag, Sizer displays a solid green "rubber band" line showing where the beam will go. Release the mouse button at the gridpoint where you wish to end the beam. Sizer draws the beam between the two gridpoints.

You can create beams between any two gridpoints. One beam can cross another beam, but two beams cannot overlap. For example, you cannot create two beams between the same pair of gridpoints. Sizer indicates an error if you try to overlap beams.

Even if a beam spans several gridpoints, it is treated as continuous. However, if you create several beams in a row, spanning a series of gridpoints along one gridline, the beams are treated as discontinuous.

Deleting Beams

To delete a beam, select it and press the DELETE key on the keyboard.

Concept Mode Data Bar

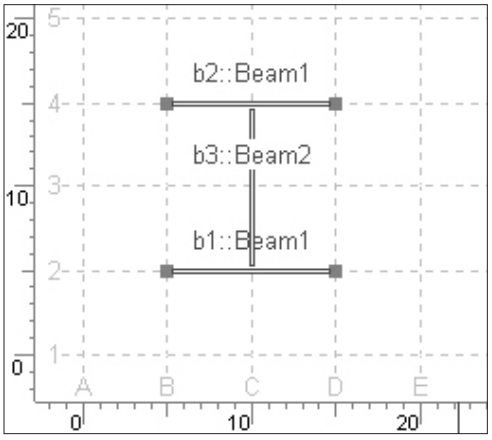
The Concept Mode Data toolbar is active in the Beam view when a beam is selected. The Concept Mode Data toolbar will display information on the selected item, including the beam name and the beam group name.

Beam Name
If a beam is selected, the Concept Mode Data toolbar will display the beam name.

As a default, the beam name is **b1** for the first beam, **b2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar.

Beam Group
If a beam is selected, the Concept Mode Data toolbar indicates which beam design group it belongs to.

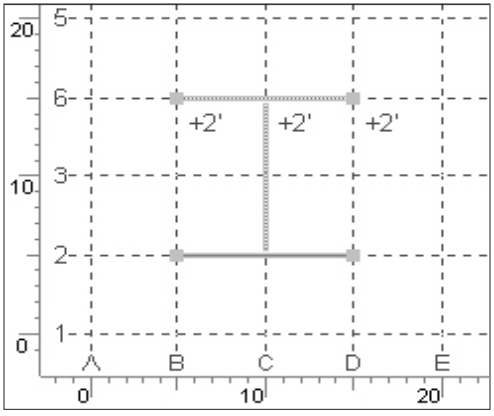
The group of a beam can be changed by selecting a different group from the drop-down list included in the toolbar.



Examples of Beam Supports in Concept Mode

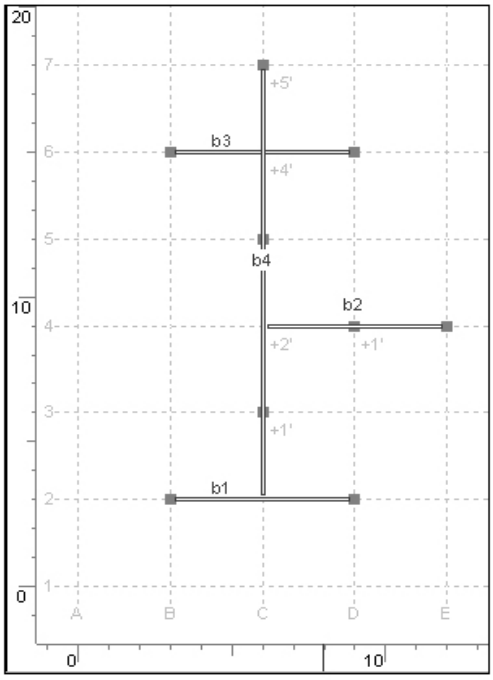
Beam supported by 2 other beams

In this example, beam group Beam1 has a load transfer number of 0. In order to create b3, its beam group, Beam2, must have a higher load transfer number than the supporting beams. In this case, a load transfer number of 1 or higher for Beam2 would work. The load transfer number is entered in the beam groups dialog (press the **Groups** button after pressing the **Beam** button on the toolbar while in Concept mode).



Sloped beam supported by 2 other beams

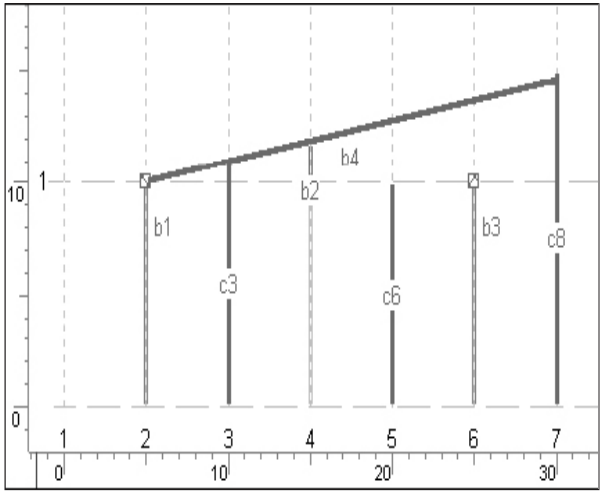
Gridpoint elevations of support points must be entered explicitly when they differ from the elevation of the floor level. From the previous example, the elevation of beam b2 has been increased by 2 feet. In order to create beam b3 it is also necessary to raise the elevation of the support point C6 to the same elevation as the rest of the supporting beam b2.



Sloped beam supported by more than 2 other beams

The slope of a beam with more than 2 possible supports is now defined by the elevation of the two extreme supports. In this example, the slope of beam b4 is defined by the elevation of beam b1 at elevation 10 feet and the column at gridpoint C7 with an elevation of 15 feet. The column at gridpoint C3 and the cantilevered end of beam b2 also support b4 because their elevations lie along the slope of b4, and, as noted in the previous example, the elevations of the support points have been entered explicitly.

Beam b3 does not support b4 even though the support gridpoint has been raised to the correct elevation. This is because the beam b3 is at a lower elevation. Its elevation is defined by the columns at B6 and D6.

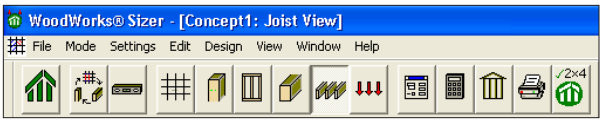


The column at C5 also does not support beam b4 because its gridpoint elevation has not been raised to meet the beam. The plan and elevation views shown here illustrates these points.

4.8 Joists

Creating Joists

Click ***joist*** on the toolbar. The ***joist*** button on the toolbar appears as light grey to indicate the Joists view. The Sizer window title also indicates the current view.



All to-be-designed joist areas appear as solid blue outlines with a blue direction indicator. All not-to-be-designed joist areas appear as solid blue outlines with a black direction indicator.

Joists
New joists must be added to existing joist groups. If you have created the joist design groups, simply choose the appropriate group from the drop-down list in the Concept Mode Data toolbar. To create a rectangular joist area, click on the four gridpoints at the corners of the area in question. As you click, a rubber-band line stretches from one gridpoint to the next. Sizer encloses the joist area defined by these four gridpoints with a solid blue outline. Within this outline is a double-headed arrow showing the direction of the joists. The arrow is blue for joists to-be-designed and black for joists not-to-be-designed.

To create a triangular joist area, click on the same gridpoint for the first and fourth points.

Joist areas can be either simply supported, multi-span or cantilevered. Joist areas must have at least two supporting beams or walls. If there are four supporting beams or walls, Sizer assumes the joists span the shorter direction. Where the number of supporting beams or walls exceeds four, you should define several joist areas, each with no more than four supporting beams or walls.

Deleting Joists

To delete a joist area, select it and press the DELETE key on the keyboard. If the joist area supports a load, Sizer does not allow you to delete it.

Concept Mode Data Bar

The Concept Mode Data toolbar is active in the Joist view when a joist area is selected. The Concept Mode Data toolbar will display information on the selected item, including the joist area name, the joist areas group name, and the joist areas direction.

Joist Area Name
If a joist area is selected, the Concept Mode Data toolbar will display the joist area name.

As a default, the joist area name is **j1** for the first joist area, **j2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar.

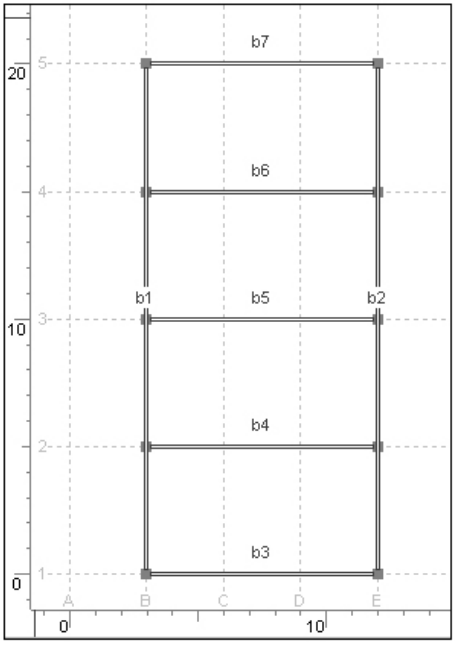
Joist Area Group
If a joist area is selected, the Concept Mode Data toolbar indicates which beam design group it belongs to.

The group of a joist area can be changed by selecting a different group from the drop-down list included in the toolbar.

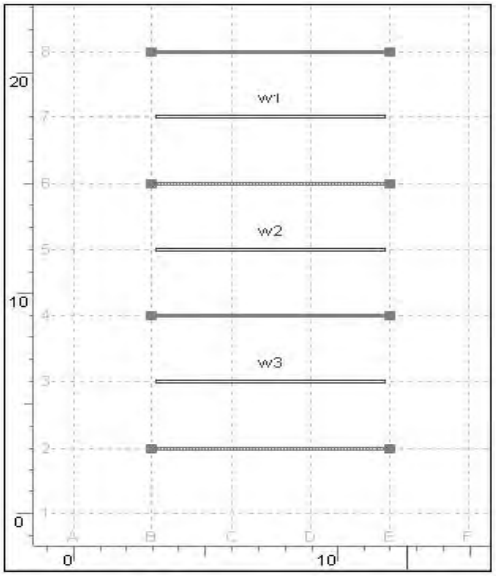
Joist Area Direction
If a beam is selected, the Concept Mode Data toolbar indicates the direction which the joist area spans.

The direction of a joist area can be changed by selecting an alternate direction from the drop-down list included in the toolbar. Note that an alternate direction is only available if there are beams or walls in the alternate direction to support the joists.

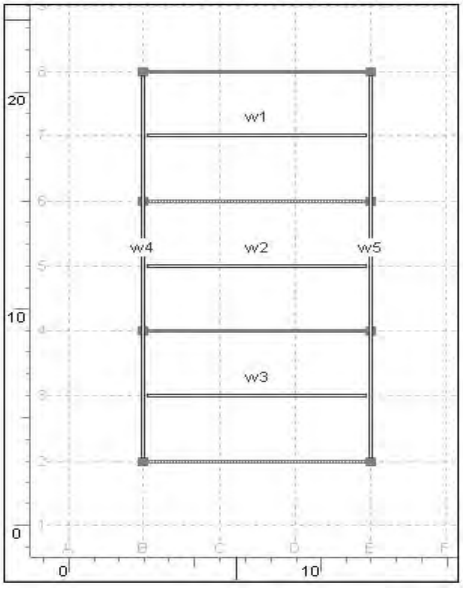
Examples for Joist Areas Sloped Joists and Gridpoint Elevations
The slope of a joist area is determined by the gridpoint elevations of the members that support it rather than by the elevations at the corners of the joist area itself. This is significant when you want to create a cantilevered joist area that sits on 2 beams at different elevations.



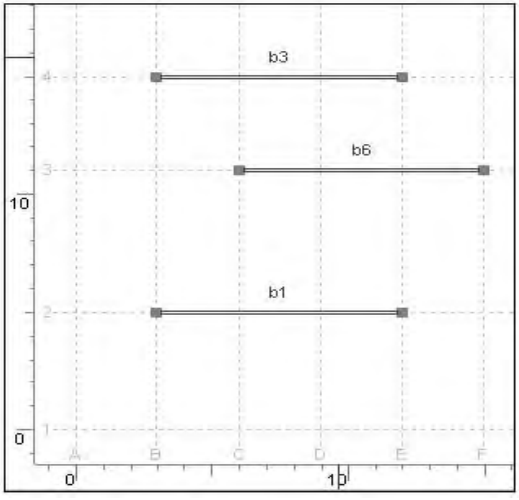
Maximum of 6 potential supports per member type
Error Message: “There are more than 6 support beams...”
Sizer is limited to considering a maximum of six potential support members of one type. In this example, an attempt to create the joist area B1-B5-E5-E1 fails because there are 5 beam supports in the N-S direction and 2 beam supports in the E-W direction, creating a total of 7 beams forming potential supports for the joist area in both directions. This example would not have failed had one of the beams been removed or replaced by a wall.



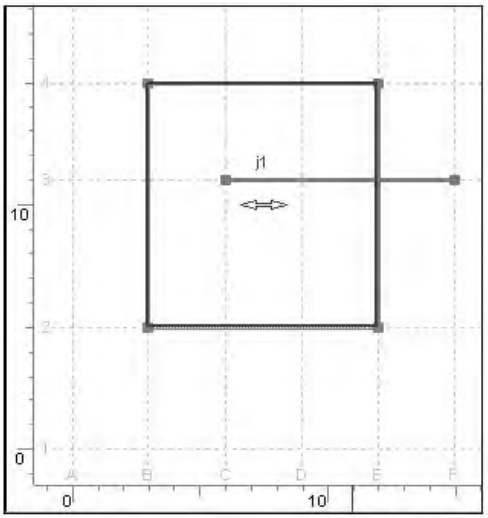
Maximum of 6 spans or 6 supports
Error Message: “No more than 6 supports or 6 spans...”
Sizer limits multi-span joists to a maximum of 6 supports or 6 spans. In this example, an attempt to create the joist area B1-B8-E8-E1 fails because there are 4 beam supports and 3 wall supports in the NS direction, exceeding the maximum of 6 supports of both types in that direction, but no supports in the E-W direction.



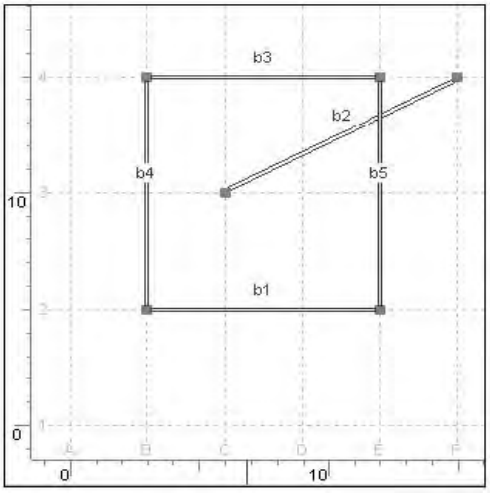
Only one valid span direction
Two walls have been added to the previous example, allowing us to create a joist area running in the E-W direction. E-W is the only feasible joist direction as there are still too many supports in the N-S direction. Note that we have not exceeded the limit of 6 beams or 6 walls in both directions.



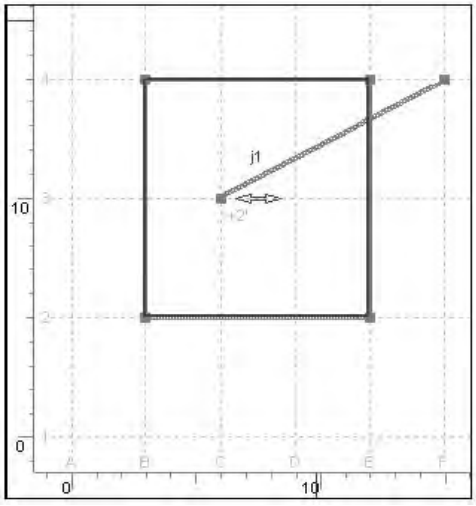
Partial Supports
Error Message: "Joist area is not FULLY spanned by at least 2 supports..."
Sizer does not permit partial supports. In this example, an attempt to create the joist area B2-B4-E4-E2 fails because beam b6 only supports some of the joists in the N-S direction, and there are no supports for joist in the E-W direction.



Partial Supports are ignored if another direction can be spanned
Two beams have been added to the previous example, running in the N-S direction. Therefore it is possible to create a joist area running in the E-W direction, even though there is a partial support disqualifying the N-S direction.

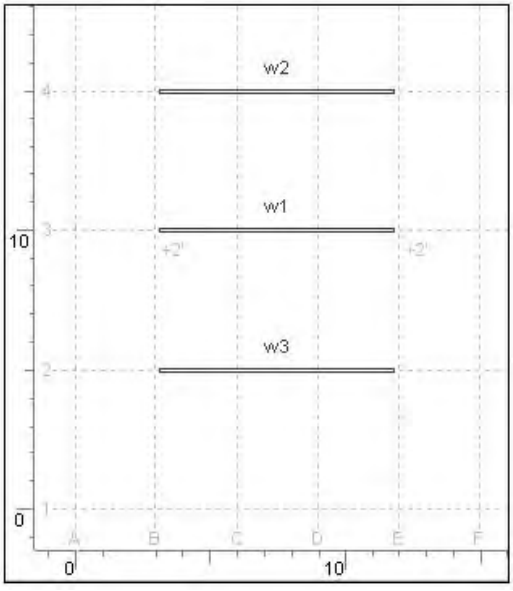


Angled Partial Supports
Error Message: "There is a beam or wall which only supports part..."
Joist areas cannot be placed over angled partial supports. In this example, the partial support is at an angle to the joists in both directions so that it supports some of the joists in both directions. This makes it impossible to create a joist area.



Partial Supports are ignored when they lie at a different elevation than the joists

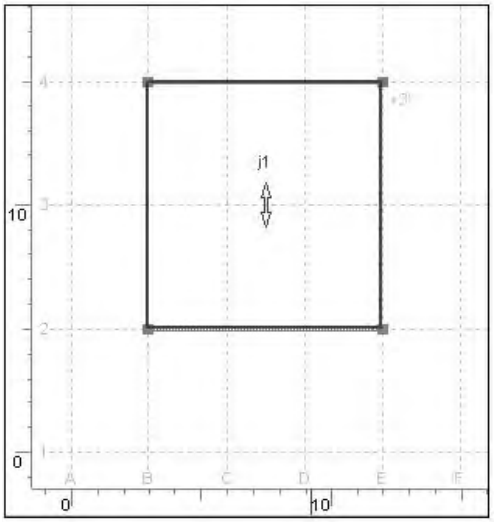
In this example, the end of the partial support has been elevated 2 feet above the other supports so that it no longer is able to support any of the joists. It is now possible to create a joist area with joists running in either direction. Sizer assumes that the column supporting the elevated beam pierces the joist area.



Out-of-plane supports

Error Message: "Joist area is not FULLY spanned by at least 2 supports..."

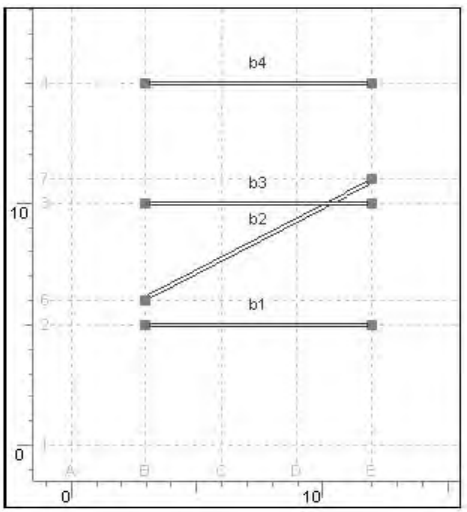
When there are more than two supports, Sizer does not permit out-of-plane supports. In this example, an attempt to create the joist area B2-B4-E4-E2 fails because all three wall supports do not lie in the same plane. A solution in this case might be to create two separate joist areas, B2-B3-E3-E2 and B4-B3-E3-E4.



Out-of-plane joist areas

Error Message: "Joist areas could not be designed because they do not lie in a plane."

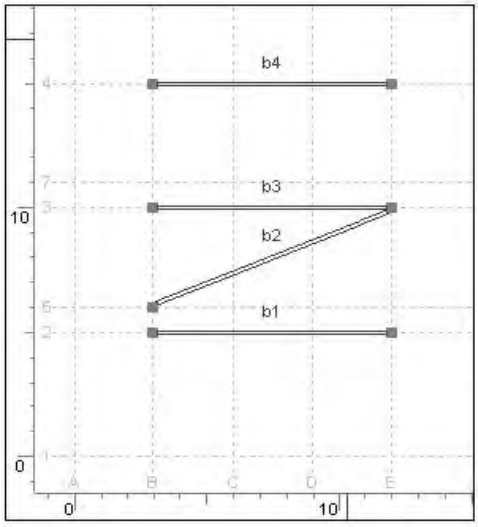
Sizer cannot design out-of-plane joist areas. Despite the fact that the two beam supports do not lie in the same plane – support B4-E4 is sloped and support B2-E2 is horizontal – the joist area B2-B4-E4-E2 can be created. However, Sizer cannot design such out-of-plane joist areas. Note that if there are more than two supports, they must all lie in the same plane.



Supports Intersect under a joist area

Error Message: "Beams or walls supporting joist area in same direction intersect."

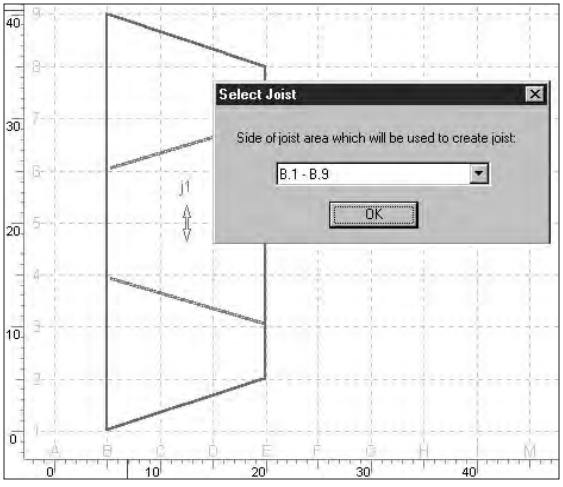
In this example, an attempt to create the joist area B2-B4-E4-E2 fails because the potential supports b2 and b3 intersect within the joist area. It is, however, possible to create a joist area that does not include the intersection; B2-B4-D4-D2 is allowed because the intersection is outside the joist area.



Supports Intersect at the edge of a joist area

Error Message: "Beams or walls supporting joist area in same direction intersect."

In this example, an attempt to create the joist area B2-B4-E4-E2 fails because potential supports b2 and b3 meet on the edge of the joist area. Not all joists would have the same number of supports if this was allowed.



Example of Transferring Joists from Concept to Beam Mode

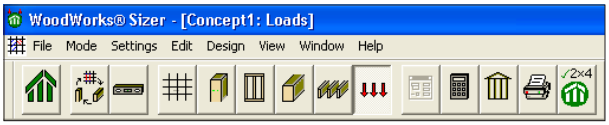
Select Joist for transfer to Beam mode

Now that Sizer permits multi-span joist areas and wall loads on joist areas, it is no longer the case that the longest joist from the joist area is critical. You are now given a choice between the two joist edges when transferring the joist to beam mode from concept mode. During the design process of the Concept mode, the edges are designed separately and the more critical side controls the design.

4.9 Loads

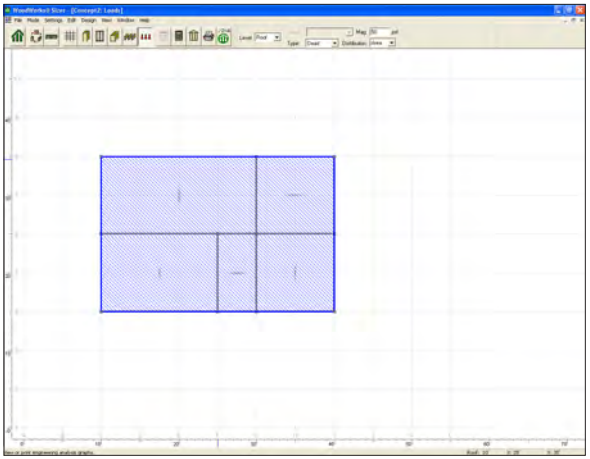
Creating Loads

Click ***load*** on the toolbar. The ***load*** button on the toolbar appears as light grey to indicate Loads view. The Sizer window title also indicates the current view.



In Loads view, area loads appear as diagonal blue lines. Area loads appear as cross-hatched diagonal lines if both live and dead area loads are applied to the same area. Line loads appear as solid blue lines in Loads view.

In Grid, Column, Wall, Beam, and Joist views, loads are not shown.



Before you create a load, you must choose its type and its magnitude. This is done through the *Concept Mode Data* Bar by selecting the load type (dead, live, snow, wind, earthquake etc.) from the **Type** drop-down list and selecting the load distribution (Line, Area or Point). Then enter the magnitude of the load as a numeric value in the field directly to the right of the **Type** drop-down list. To load a rectangular area, click on the four gridpoints at the corners of the area in question. As you click, a rubber-band line stretches from one gridpoint to the next. Sizer fills the load area defined by these four gridpoints with diagonal blue lines.

Since a structure may not be rectangular, Sizer allows you to create a load area larger than the structure. The load is automatically applied only to the joists contained within the selected area.

To create a uniformly distributed line load, double-click a beam. The loaded beam is displayed in blue. Load walls the same way.

You can create any number of overlapping loads.

Deleting Loads

To delete an area or beam load, select it and press the DELETE key on the keyboard.

Concept Mode Data Bar

The Concept Mode Data toolbar is active in the Load view when a load is selected. The Concept Mode Data toolbar allows you to edit and displays information on the selected load. This includes the load name, type, and magnitude.

Note: To select a load, click on the load. Selected loads appear in red. If loads overlap, simply click repeatedly until you reach the desired load.

Load Name

If a load is selected, the Concept Mode Data toolbar will display the load name.

As a default, the load name is **load1** for the first load, **load2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar.

Load Type

If a load is selected, the Concept Mode Data toolbar will display the load type. This drop-down list contains the types of loads available (dead or live).

Note: Live loads applied on the roof level are automatically treated as Snow loads. To specify other load types for roofs (e.g. Wind), change the load duration factor for the Snow load type in the Load Duration Factors dialog box(U.S. only).

Load Magnitude

If a load is selected, the Concept Mode Data toolbar will display the load type. This field allows you to assign the magnitude for an area load (psf or kN/m^2) or for a line load (plf or kN/m).

Loads and Load Combinations

Load Direction
All live and dead loads are applied vertically downward.

Load Location
Sizer automatically applies dead loads along the length of a sloped member. Live loads are applied along the horizontal projected length of a sloped member.

Load Combinations
Combinations are automatically generated for the following loads:

- dead load only
- dead and live loads

Pattern Loads
Pattern loads are only available in Beam mode for live, snow, wind, construction (U.S. only) and impact loads (U.S. only) on multi-span beams.

Load Transfer
Loads are transferred in Concept mode using the following rules:

- Joists transfer to supporting beams or walls.
- Beams transfer to supporting beams or columns.
- Walls transfer to supporting walls or beams on a lower floor.
- Columns transfer to columns or beams on a lower floor.

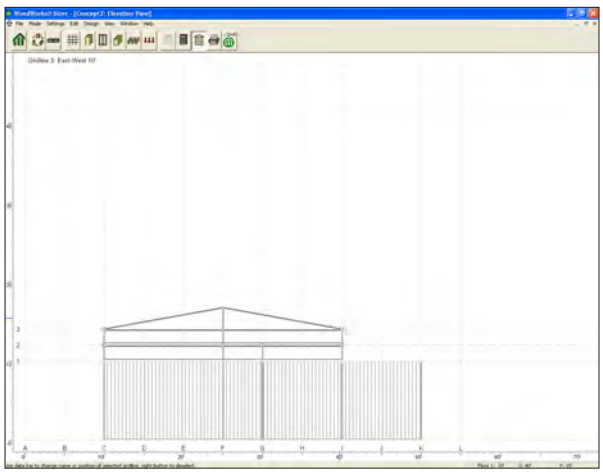
Sizer determines whether a beam is supporting or supported based solely on the Load Transfer Number (LT#) entered for the beam's design group. A beam with a higher LT# always transfers loads to beams with lower LT#'s. In other words, a beam with a lower LT# always supports a beam with a higher LT#.

There are two points to note, however:

- Sizer determines maximum joist and beam reactions from a structural analysis of each member. In the case of joist areas, Sizer only analyzes the first and last joists by determining the reactions at the two supports of the first and last joists. Sizer uses these computed reactions as the start and end values of a distributed load on the supporting beams or walls. It is assumed that the resulting distributed load varies linearly between the first and last joists. In the case of a rectangular joist area on parallel supports, this produces a uniformly distributed load on the beam or wall. Any other conditions produce a trapezoidal load on the beam or wall. Should uplift occur at one end of a joist area but not the other, Sizer computes two triangular distributed loads—a positive load over a portion of the supporting member and a negative load over the remainder of the member.
- Walls supported by beams on a lower floor cannot span over a column unless the beam is continuous over the column.

4.10 Elevation View

Sizer includes a feature to view the elevation or cross-section of your structure along any gridline. To do this, change to the Grid view and select any gridline in either the North-South or East-West directions. The selected gridline will appear in red. Now click on the ***elevation view*** button and the program automatically displays the elevation view along the selected gridline.



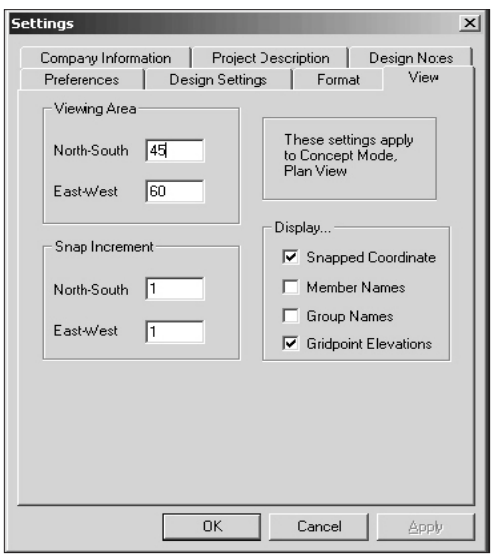
Member Display

The elevation view displays members in the following fashion:

- Members that are on the selected gridline are shown in red.
- Members that are not on the current gridline and are South or East of the gridline are shown in light grey lines.
- Members that are not on the current gridline and are North or West of the gridline are shown in dark grey lines.

View Options

The elevation view can be enhanced to include the member names and the group names by selecting these options in the ***View*** tab from the ***Settings*** dialog.

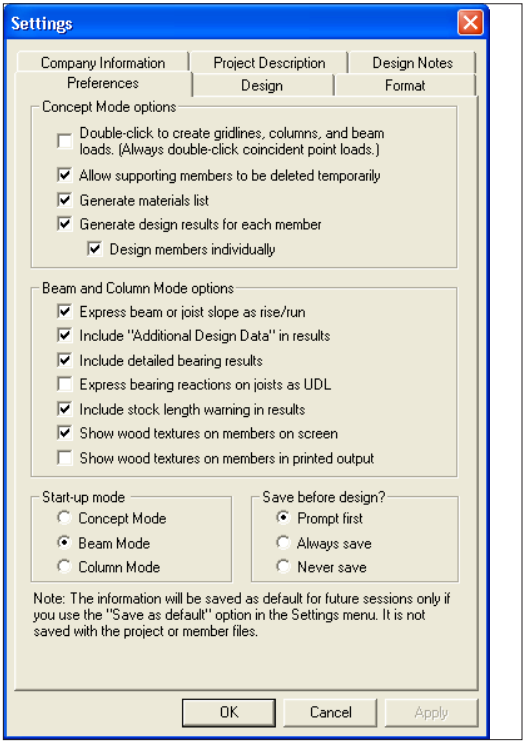


4.11 The Design Process

Starting the Design Process

To start the design process, click ***design*** on the toolbar. Sizer performs an analysis and a design of your structure with the information you entered and automatically displays the results.

Sizer normally prompts you to save the current project prior to doing the design. To change this, choose one of the ***Save Before Design*** options from the ***Preferences*** tab under the ***Settings*** dialog.



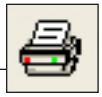
Limiting the Design

By default, Sizer designs the elements for your entire project. However, you can force Sizer to design only the elements on the current level.

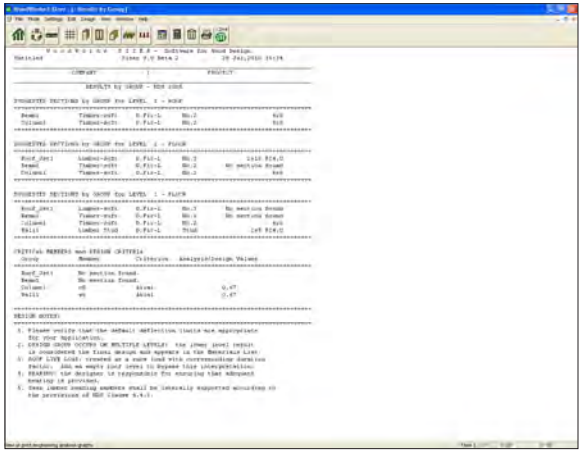
To limit the design, choose ***Design Current Level Only*** from the ***Preferences*** item under the ***Settings*** menu.

To design the elements for the entire project, choose ***Design Entire Structure*** from the ***Preferences*** tab under the ***Settings*** dialog.

Viewing and Printing the Results



Once Sizer has designed the elements in your project, it automatically displays the design summary on the screen.



Use the scrollbars (or the PgUp and PgDn keys) to scroll though the results.

To close the results window, double-click the close box in the top left corner of the window.

To redisplay the design results at any time, choose ***Design Results*** from the ***View*** menu.

To print the results, choose ***Print*** from the ***File*** menu. The Print sub-menu provides several options (such as Suggested Sections).

There are new files and diagrams available for printing in Concept mode.

Reactions at Base drawing shows that Column and Wall reactions at the base of a structure.

Results by Member shows the design response of all the members in each design group.

Design Members Individually shows the design of each member in the entire structure.

Materials List tabulates all the material used in the structure.

4.12 Advanced Features

Performing a Detailed Design

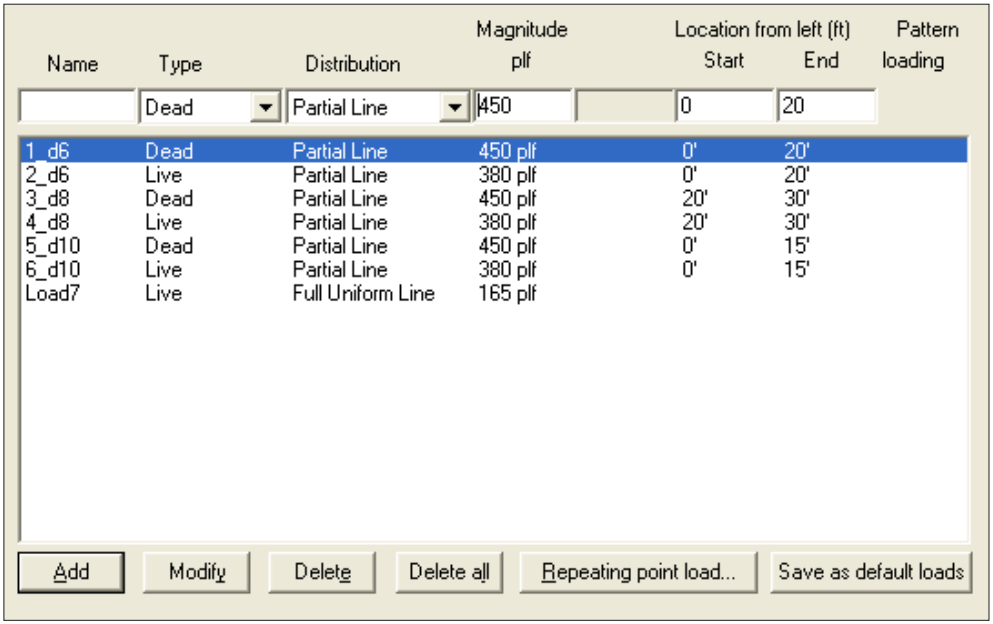
The Concept mode design process computes the lightest acceptable section size for each design group. However, these results show only a limited amount of detail, and they do not indicate the margins by which the sections passed the various criteria (bending, shear and deflection).

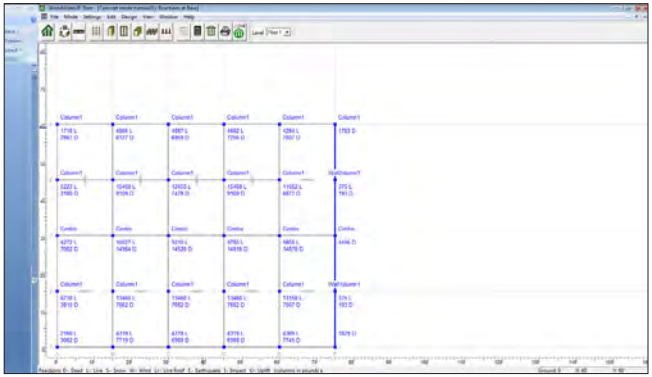
Sizer can perform a detailed design for single beams, joists, columns and walls. This is done by importing the critical member into the Beam or Column mode. To import, click on the critical member so that it appears in red, and then press the ***mode*** button on the main toolbar. This imports the member into the appropriate single member mode for a more detail design and analysis.

Sizer first determines the reactions of the other members to be transferred to the selected one, and converts the reactions to the appropriate types of loads. Sizer then transfers these loads (along with all the design group information) to Beam or Column mode, where you can perform a detailed design or code check on the member.

The load names of these transferred loads indicate where each of the loads are coming from. For example, a load named *1_j1* would come from joist area *j1*.

Important note: You cannot transfer a member from Beam mode or Column mode back to Concept mode.

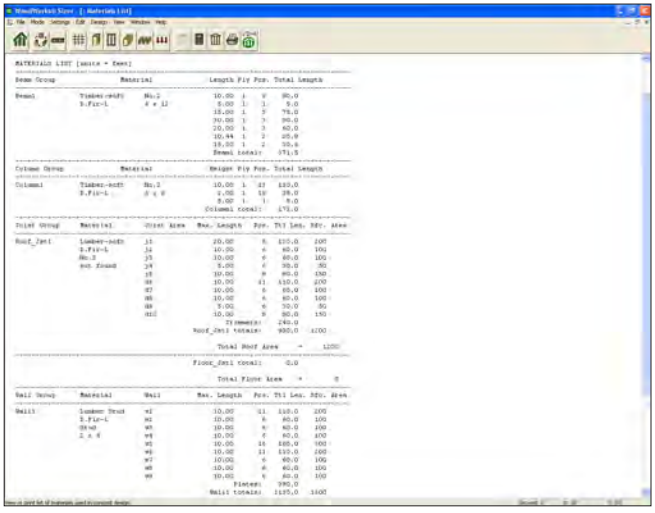




Reactions at Base

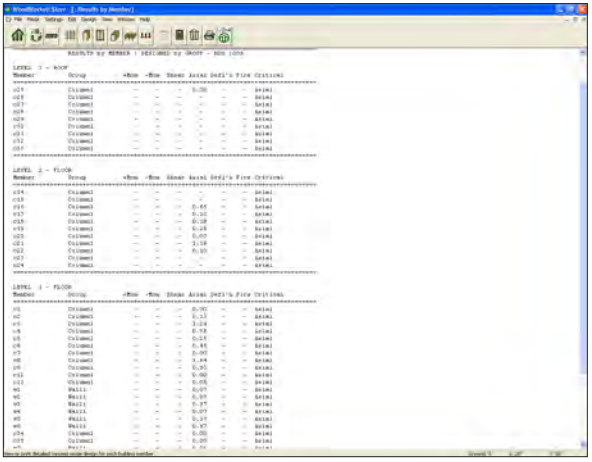
Select this option from the View menu (or from the right-button context menu) to display wall and column reactions at the base of the structure. Wall loads are output as line loads (plf or kN/m) and columns loads as point loads (lbs or kN).

Due to the way Sizer designs walls it is possible that, for some layouts, the sum of the reactions will exceed the sum of the applied loads. This occurs when the intensity of loads applied to the top of a wall is not uniform along the entire wall length. Sizer designs walls by determining the largest load on a single wall stud within the wall. This same design load becomes the reaction that is transferred to a lower story as a uniform load along the entire length of the wall. This is a conservative approach since the single largest load at the top becomes the uniform reaction at the bottom.



Materials List

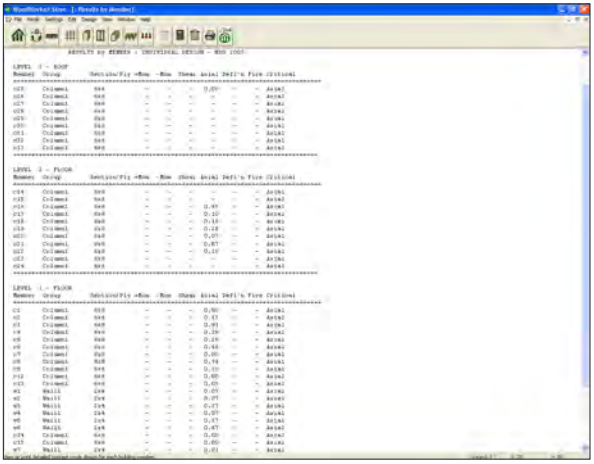
Select this option from the View menu (or from the right-button context menu) to display a list of materials. All the materials used in the structure are listed in a table, organized by group. The table includes the number of pieces for each length of the material, area of floors, roofs and walls, total length of joist trimmers and wall plates.



Results by Member

Designed by Group

Each member is listed in a table that displays how it performed with respect to various design criteria (bending, shear, deflection, axial, and fire endurance). The information in this table can be used to evaluate which members in a group are the least or most "stressed". You can disable this feature in the Settings Preferences tab. Activating this feature increases the time it takes Sizer to complete a design.



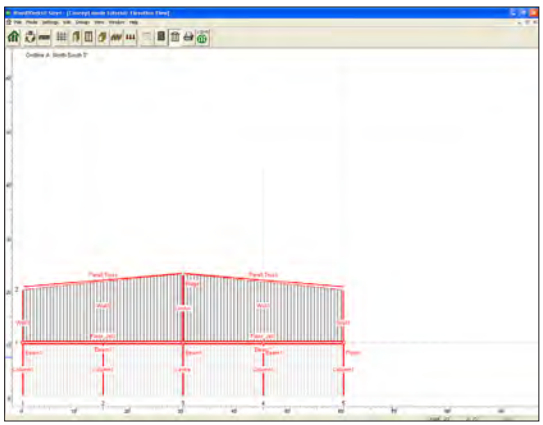
Individual Design

Sizer includes an option that allows you to design each member of the structure individually rather than designing using the worst case member in the design group. You can enable this feature through the **Settings Preferences** tab by checking the ***Design members individually*** box. This option will modify the Results by Member report to include the design for each member along with information on how each member performed.

4.13 Tutorial

Introduction

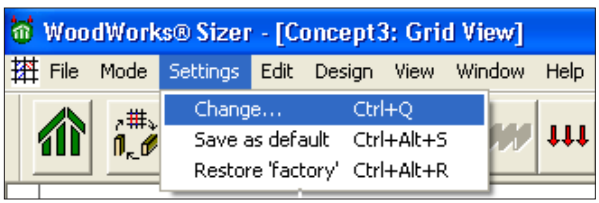
In this tutorial you will create and design a commercial two-story wood-frame structure with a pitched roof as shown (in cross-section) below.



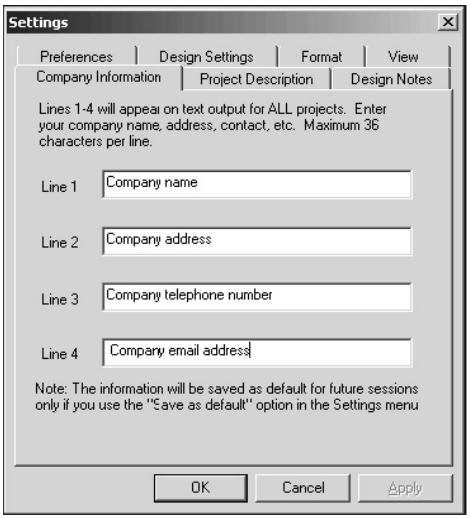
To begin, ensure that you are in the Concept mode by selecting ***Concept*** from the ***Mode*** menu.

Company Information

1. From the ***Settings*** menu under ***Change***, choose the ***Company Information*** tab.

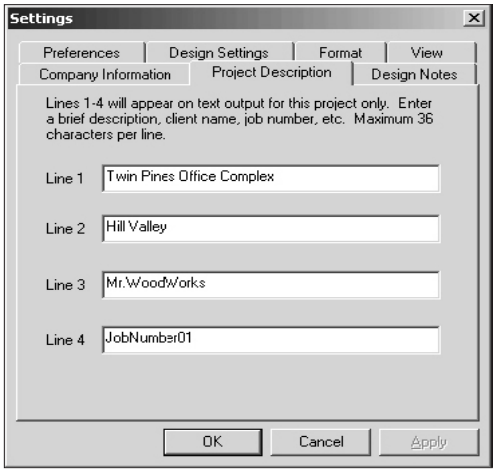


2. Enter relevant company information.



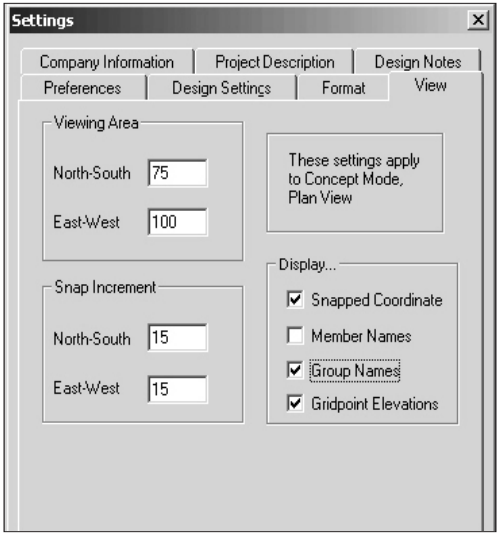
Project Description

1. Now click on the ***Project Description*** tab.
2. Enter relevant project information.



Snap Increment and Display Options

1. Click on the ***View*** tab.
2. In the ***Viewing Area*** field, increase the ***North-South*** limit to **75.0**(ft) and the ***East-West*** limit to **100.0**(ft).
3. Enter ***North-South*** and ***East-West Snap Increments*** of **15.0**(ft).
4. Switch the ***Display... Group Names*** option to on, so that an 'x' appears in this field.



5. Click ***OK***.
6. (Optional) To save these settings as default settings, select ***Save New Settings*** under the ***Settings*** menu.

Levels Above Grade

1. Click on the ***levels*** button from the main toolbar.

2. Enter a roof elevation of **20**(ft). Click ***Add***. Click ***OK***.
3. Click on ***Floor 1*** so that it is highlighted. Click ***OK***. (This will change the current level to ***Floor 1***).

Gridlines

1. Click on the ***grid*** button from the main toolbar.

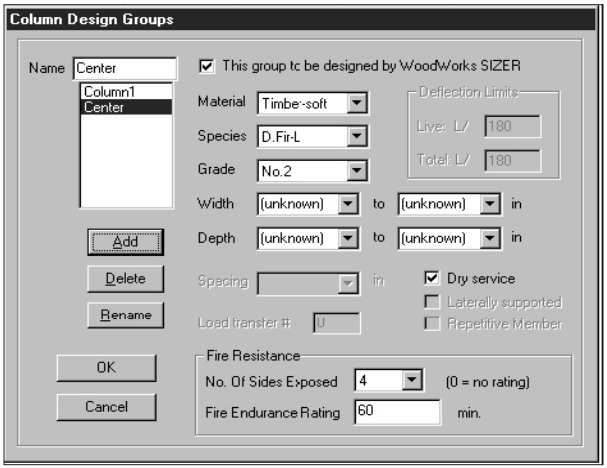
2. Click at the following X and Y locations on the grid:

X = 0(ft)	Y = 0(ft)
X = 15(ft)	Y = 15(ft)
X = 30(ft)	Y = 30(ft)
X = 45(ft)	Y = 45(ft)
X = 60(ft)	Y = 60(ft)
X = 75(ft)	Y = near edge of scale

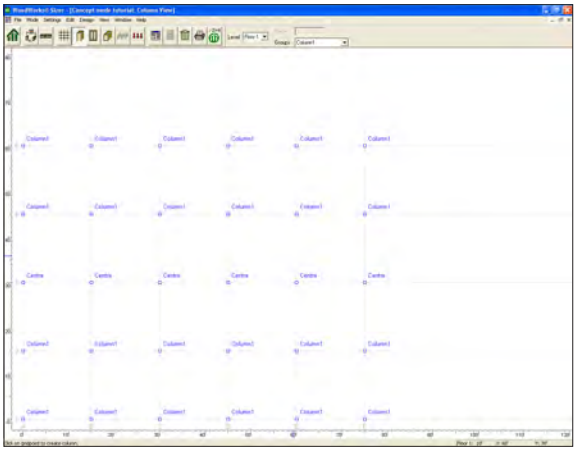
Floor Level

Columns

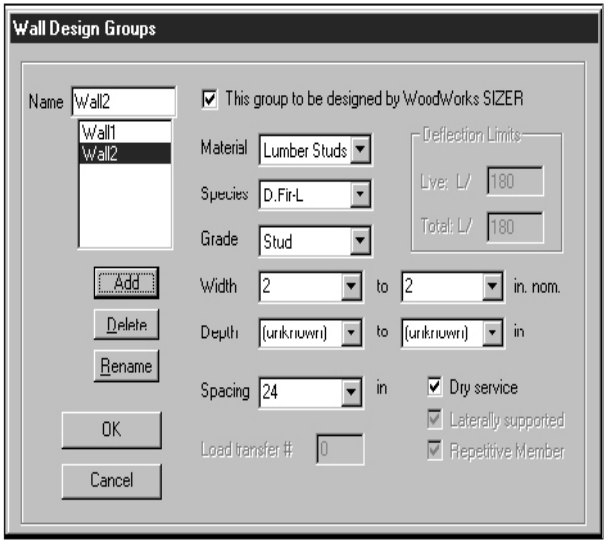
1. Click ***column*** on the toolbar.
2. Click ***Design, Design Groups*** to define an additional column group.



3. Enter a new name in the ***Name*** field: ***Centre***.
4. Click ***Add***.
5. Click ***OK***.

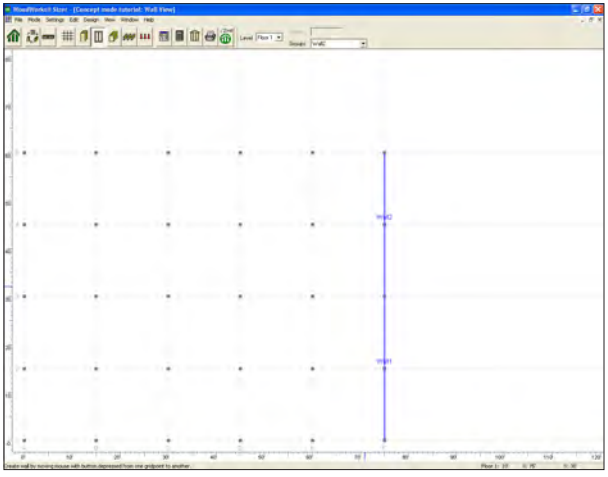


6. Choose ***Centre*** from the ***Group*** drop-down list on the data bar.
7. Click gridpoints **A-3**, **B-3**, **C-3**, **D-3**, **E-3**, and **F-3**.
8. Choose ***Column1*** from the ***Group*** drop-down list on the data bar.
9. Click all remaining gridpoints.

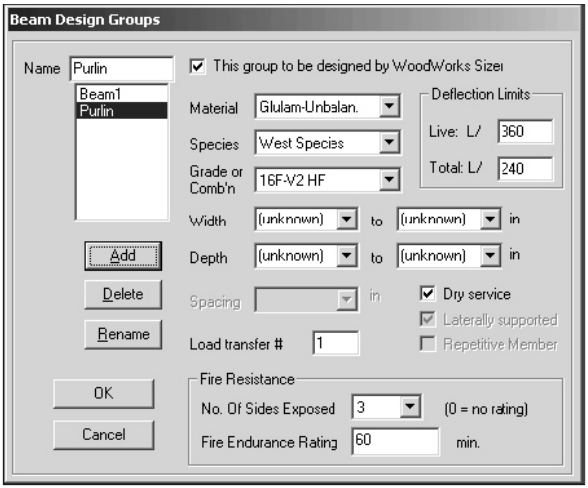


Walls

1. Click ***walls*** on the toolbar.
2. Click ***Design, Design Groups*** to define an additional wall group.
3. Enter a new name in the ***Name*** field: ***Wall2***.
4. Click ***Add***.
5. Click ***OK***.

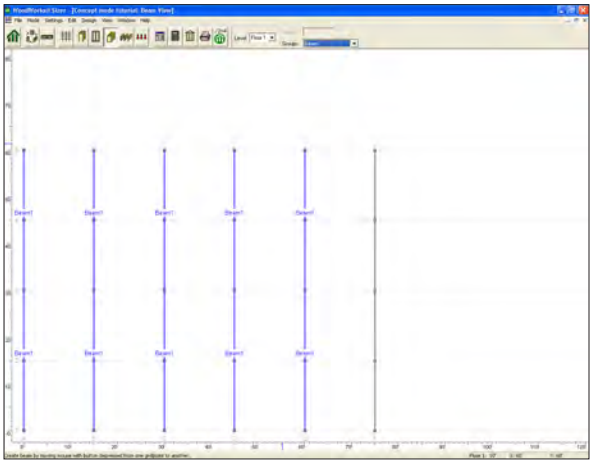


6. Choose ***Wall1*** from the ***Group*** drop-down list on the data bar.
7. Point to gridpoint **F-1**, click and drag a wall to **F-3**.
8. Choose ***Wall2*** from the ***Group*** drop-down list on the data bar.
9. Point to gridpoint **F-3**, click and drag a wall to **F-5**.

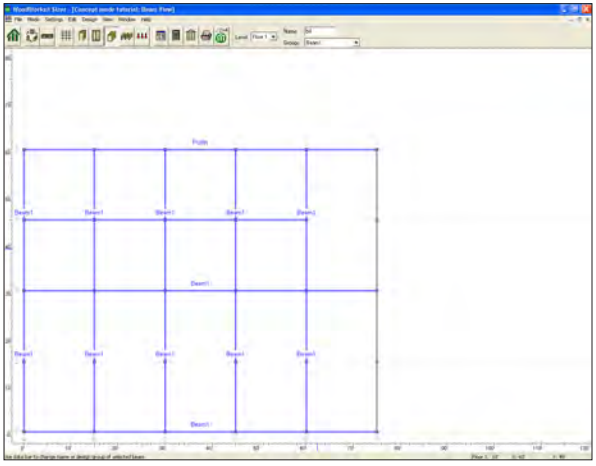


Beams

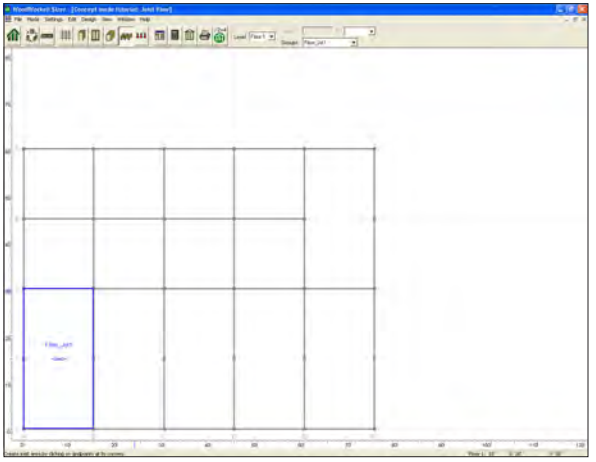
1. Click ***Beam*** on the toolbar.
2. Click ***Design Groups*** to add a beam group.
3. Select the ***Glulam Unbalan*** material type in the ***Material*** field.
4. Select the ***Name*** field and enter a new name: ***Purlin***.
5. Enter a value of **1** in the ***Load Transfer*** box.
6. Click ***Add***.
7. Click ***OK***.



8. Point to gridpoint **A-1**, click and drag a beam to **A-3**.
9. Repeat step 8 to create beams spanning in the North-South direction as shown on the left screen.



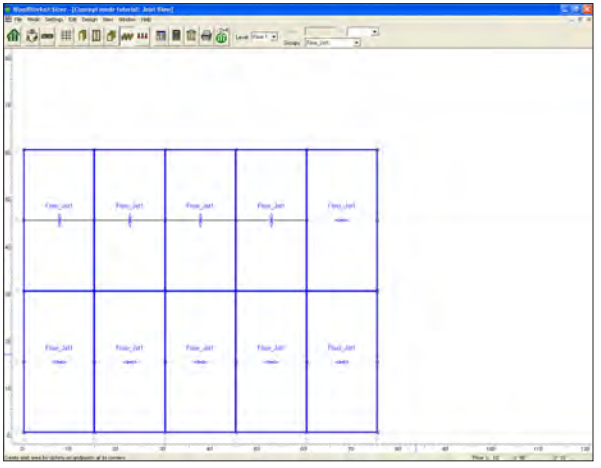
10. Choose ***Purlin*** from the ***Group*** drop-down list on the status bar.
11. Point to gridpoint **A-1**, click and drag a beam to **B-1**.
12. Repeat step 11 to create single span beams spanning in the East-West direction as shown on the left screen.



Joists

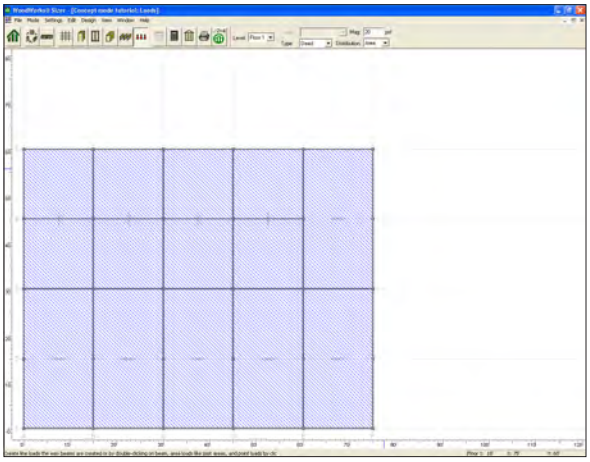
1. Click on the ***Joist*** button.
2. Click gridpoints **A-1**, **B-1**, **B-3** and **A-3** to create the first joist area.
3. Repeat step 2 to create the joist areas as shown above.
4. Click gridpoints **E-5**, **E-4**, **D-4** and **D-5**.

Note: In this case, the joists could span either North-South or East-West. You can change the direction you wish the joists to span by highlighting the joist area and then changing the direction indicated in the ***Direction*** field of the data bar.



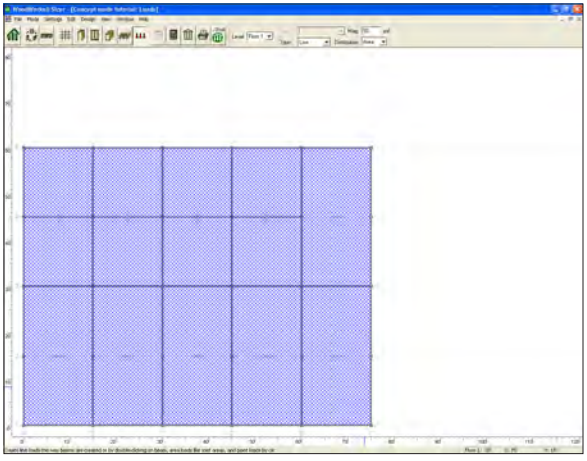
5. Repeat step 4 to create the remaining joist areas as shown above.

Note: The Concept Mode Data bar can be moved to a different location on screen for ease of use by clicking on the Data Bar and while holding the mouse button down, dragging the Data Bar around the screen.

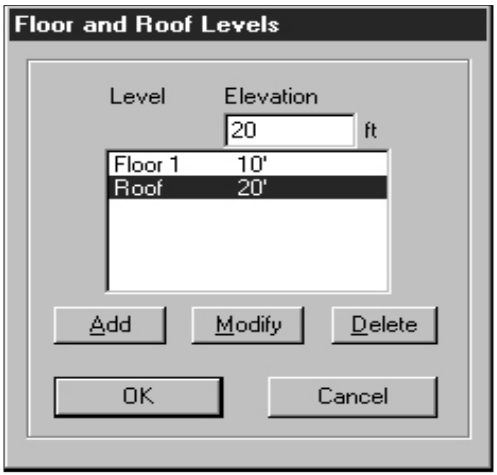


Loads

1. Click ***Load*** on the toolbar.
2. The ***Load Type*** should be set to ***Dead Area***. Enter a load magnitude of **20 (psf)** in the right most field of the data bar.
3. Click gridpoints **A-1**, **F-1**, **F-5**, and **A-5** to load the area.



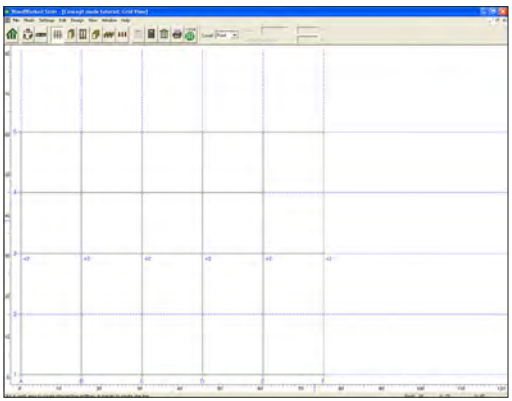
4. Choose **Live Area** from the **Load Type** drop-down list on the data bar.
5. Enter a load magnitude of 50 **(psf)** in the data bar.
6. Click gridpoints **A-1**, **F-1**, **F-5**, and **A-5** to load the area.



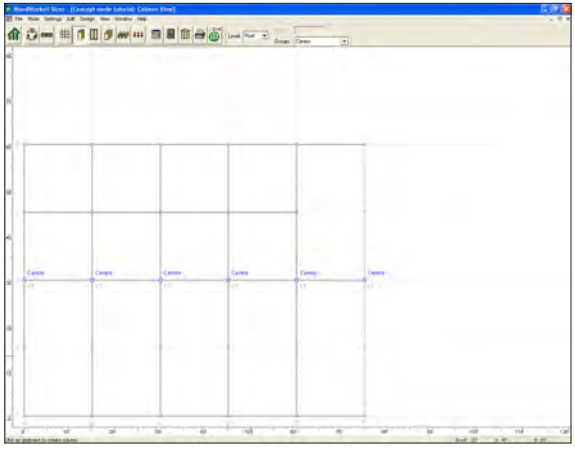
Roof Level

Gridlines

1. Click the right mouse button. Change the current level to ***Roof*** by clicking on ***Up One Level***.

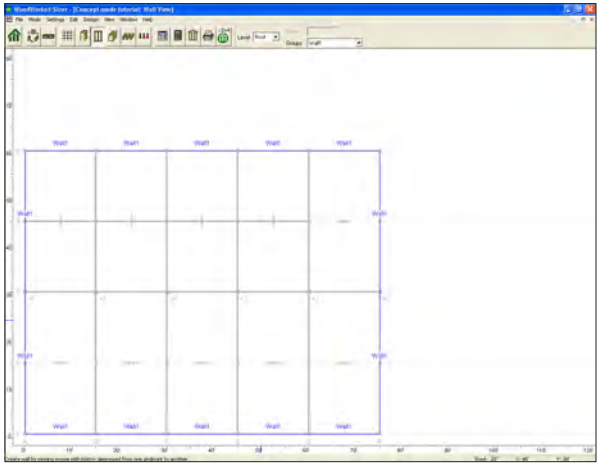


2. Click on ***Grid*** from the toolbar.
3. Select gridpoint **A-3** so that it is highlighted in red (both gridline **A** and **3** should appear in red).
4. Select the ***Gridpoint Elevation*** field from the data bar and enter an elevation of **23** (ft) and press **Enter**.
5. Create elevations of **23** ft at the following locations:
   B-3
   C-3
   D-3
   E-3
   F-3



Columns

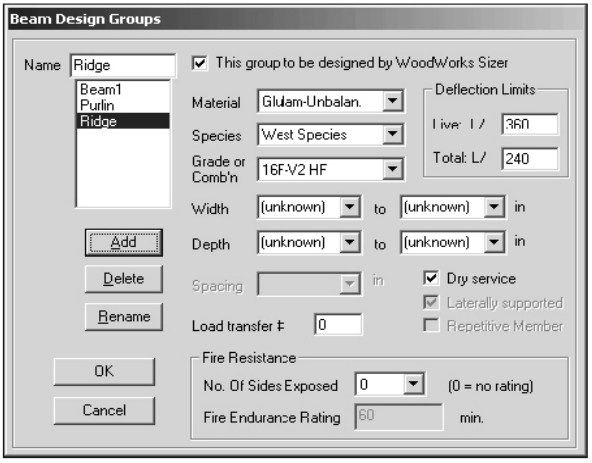
1. Click ***Column*** on the toolbar.
2. Choose ***Centre*** from the ***Group*** drop-down list on the data bar.
3. Click gridpoints **A-3**, **B-3**, **C-3**, **D-3**, **E-3**, and **F-3**.



Walls

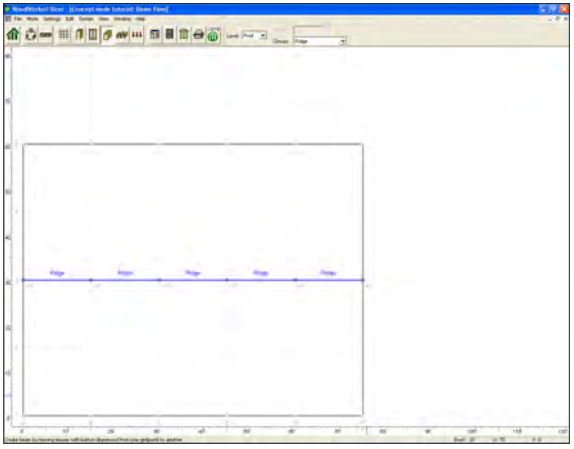
1. Click ***Wall*** on the toolbar.
2. Choose ***Wall1*** from the ***Group*** drop-down list on the data bar.
3. Point to gridpoint **F-1**, click and drag a wall to gridpoint **F-3**
4. In a similar manner, create the remaining walls as shown on the left screen.

Note: You cannot define a continuous stud wall for either the North or South walls since the beams that support these walls from below are not continuous over the columns.

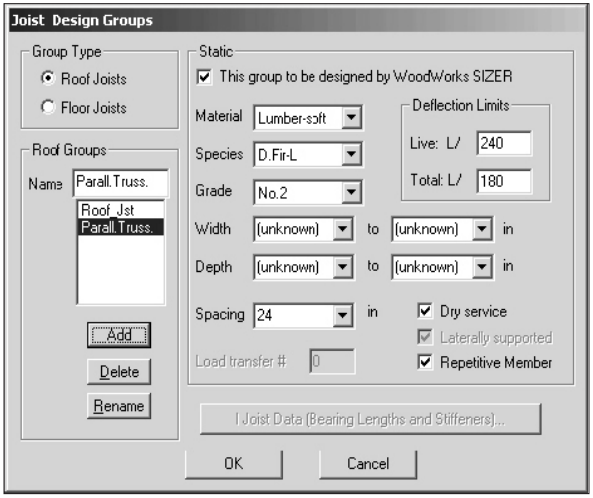


Beams

1. Click ***Beam*** on the toolbar.
2. Click ***Design Groups*** to define an additional beam group.
3. Select the ***Name*** field and enter a new name: ***Ridge***.
4. Choose ***Glulam Unbalan*** from the ***Material*** drop-down list.
5. Click ***Add***.
6. Click ***OK***.
7. Choose ***Ridge*** from the ***Group*** drop-down list on the data bar.

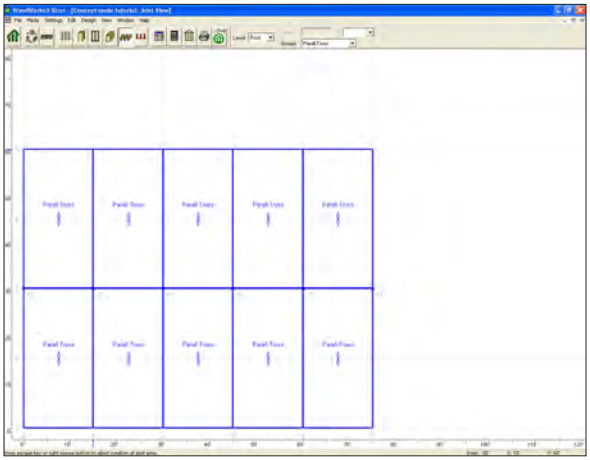


8. Point to grid point A-3, click and drag a beam to **B-3**.
9. Repeat step 8 to create beams between:
   B-3 and C-3
   C-3 and D-3
   D-3 and E-3
   E-3 and F-3

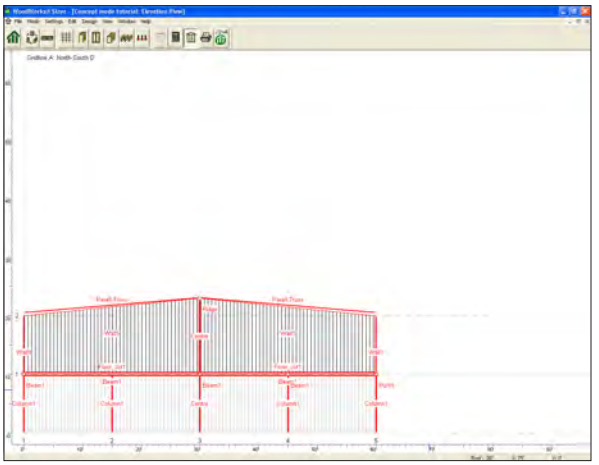


Joists

1. Click ***Joist*** on the toolbar.
2. Click ***Design Groups*** to define an additional joist group.
3. Select ***Roof Joist*** in the ***Type*** field.
4. Select the ***Name*** field and enter a new name: ***Parall.Truss.***
5. Click the check box beside ***To Be Designed***. (This tells Sizer not to size the members belonging to this group. Use this feature to model trusses or non-wood load-bearing members in the structure.)
6. Click ***Add*** and then click ***OK.***
7. Choose ***Parall.Truss*** from the ***Group*** drop-down list in the status bar.

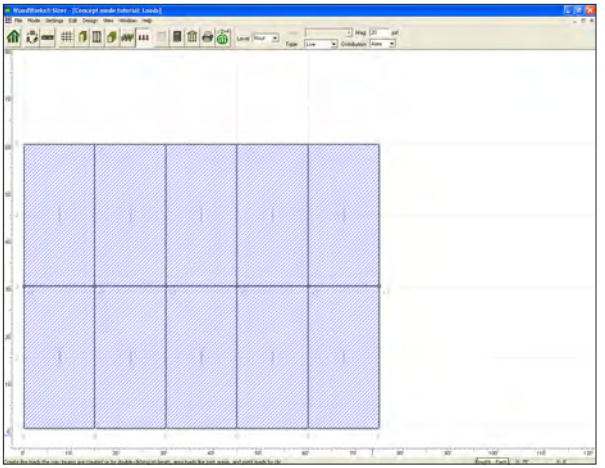


8. Click gridpoints **A-1**, **B-1**, **B-3**, and **A-3** to create the first joist area.
9. Repeat step 7 to create the remaining joist areas as shown on the left screen.



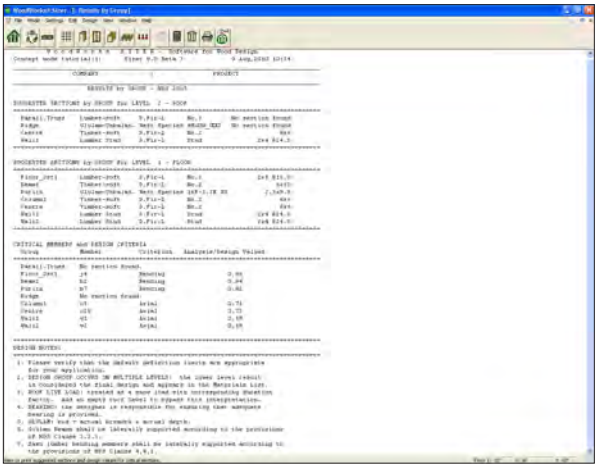
Elevation View

1. Click on the ***Grid*** button from the toolbar.
2. Click on gridline **A** so that it is highlighted in red.
3. Click on ***Elev. View*** from the toolbar.
4. The elevation view along the North-South gridline "A" is now shown.
5. To see successive views along gridlines B, C, D, etc. click the ***Grid*** button, and repeat Steps 1 and 2.
6. To return to the main window, click any of the toolbar buttons.



Loads

1. Click ***Load*** on the toolbar.
2. Choose ***Dead Area*** from the ***Type*** drop-down list on the data bar.
3. Enter a load magnitude of **20** (psf) in the data bar.
4. Click gridpoints **A-1**, **F-1**, **F-5** and **A-5** to load the area.



Design the Members

1. Click ***Design*** on the toolbar. Sizer designs all of the members in your structure and then displays the results.

View Results

1. Click the up or down arrows to scroll through the results.
2. To close the results window, click on any of the view toolbar buttons.

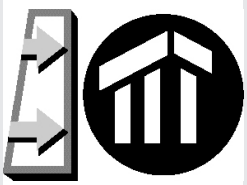
More Practice

To further familiarize yourself with Concept mode, try the following:

1. Move some gridlines and click ***Design*** on the toolbar to re-design the structure.
2. Try using transfer beams (columns supported on a beam).
3. Create an addition to your project.

What is Shearwalls?

B.1



1.1 About Shearwalls

Shearwalls is an engineering design aid for lateral wind and seismic load determination and for the design of shearwalls for buildings subject to these loads. Shearwalls will:

- Import CAD .wmf files to establish the building footprint for each level
- Model wood structures with up to six stories
- Automatically calculate seismic and wind loads based on the building code, relevant standards and site information
- Accept user specified seismic and wind loads applied to building surfaces
- Accept input for dead loads and wind uplift loads acting on shearwalls
- Distribute shear forces to shearwalls using either a flexible or rigid diaphragm analysis (including torsional effects)
- Accept input of forces applied directly to shearwalls to adjust the shearline load distributions
- Allow the specification of openings in shearwalls, such as windows and doors
- Design shearwalls for their sheathing thickness and nailing requirements
- Determine the capacity of wall sheathing and nailing to resist wind suction
- Specify required hold-down forces for shearwall segments
- Specify required dragstrut forces across openings and at the ends of shearwalls
- Design standard shearwalls with hold-downs at each segment, perforated shearwalls (U.S. only) or shearwalls with anchorages in lieu of hold-downs (Canada only).

Getting Started Button – Provides instructions on the main steps required to design a building using Shearwalls.

1.2 Design Methods

U.S. Version

Codes and Standards referenced: International Building Code (IBC®), Minimum Design Loads for Buildings and Other Structures (ASCE 7), National Design Specification for Wood Construction (NDS®), and AF&PA's Special Design Provisions for Wind and Seismic (SDPWS).

Design Procedure: Allowable Stress Design (ASD)

Wind Procedure: ASCE 7 Method 2 (Analytical Procedure), both All-heights and Low Rise

Seismic Procedure: ASCE 7 Equivalent lateral force procedure

Horizontal Distribution: Flexible and rigid diaphragm distribution

Shearwall types: Individual full-height wall segment and perforated shear walls. Includes deflection and story drift analysis.

Canadian Version

Codes and Standards referenced: National Building Code of Canada, User's Guide - Structural Commentaries, CSA O86 Engineering Design in Wood (Limit States Design).

Wind Procedure: NBCC High-rise (fig I-15) and NBCC Low-rise (fig I-7/8)

Seismic Procedure: NBCC Equivalent static force procedure

Horizontal Distribution: Flexible and rigid diaphragm distribution

Shearwall types: Segmented with hold downs and segmented with hold downs removed

1.3 Input and Output

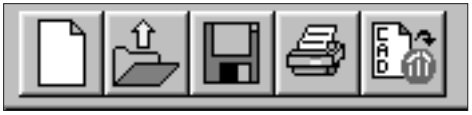
Shearwalls generates several file types to store general program settings and project information for a design run. This includes the following file types:

- **.wsw** – main file type for Shearwalls which stores binary project data and can be opened directly by Shearwalls to retrieve previously saved runs.
- **.pdf** – Engineering design results can be saved as Portable Document Format (.pdf) files located in the same folder as the corresponding .wsw file. These can then be opened by Adobe Acrobat or another pdf reader.
- **.rtf** – Engineering design results can be saved as Rich Text Format (.rtf) files located in the same folder as the corresponding .wsw file. These can then be opened and viewed by text editors such as Word and Wordpad, and converted to Word document (.doc) files.
- **.log** – intermediate calculations used to generate wind and seismic loads, and used in the rigid diaphragm analysis are stored in text files with the same name and located in the same folder as the corresponding .wsw file. These files can be opened and viewed through any text editor, such as Notepad or Word.
- **shearwalls.wss** – *Standard Walls* are stored in this binary file located in the same folder as the Shearwalls program. If lost, it will be regenerated automatically by Shearwalls.
- **Shearwalls.ini** – Program settings that apply to all designs are stored in this Initialisation file, located in the Shearwalls installation folder.

1.4 Toolbars

The toolbar across the top of the main window is split into three groups of buttons:

- **File Operations:** ***New, Open, Save, Print, Import***



- **Actions:** ***Structure, Walls, Openings, Extend Walls, Roofs, Building Site, Generate Loads, Loads and Forces, Design***

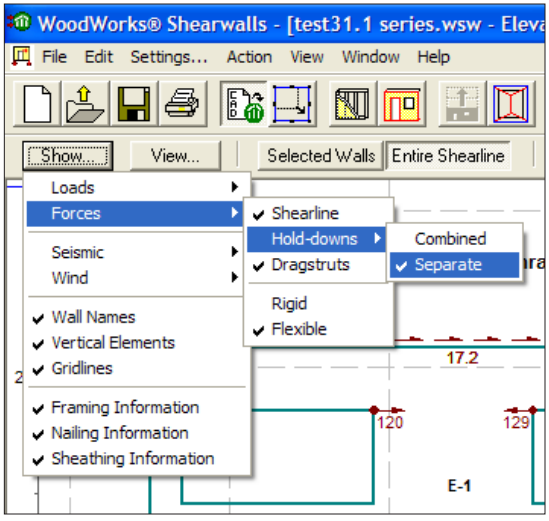
- **Views:** ***Plan View, Form View, Elevation View, Results View***

For both the ***Action*** and the ***View*** buttons, a depressed button indicates the current activity or view.

Status Bar
The bar at the bottom of the screen displays a line of text that contains hints and instructions to the user, and descriptions of the programs buttons, menu items, and data fields. Consult the status bar when you are uncertain as to how to proceed.

1.5 Data Bars

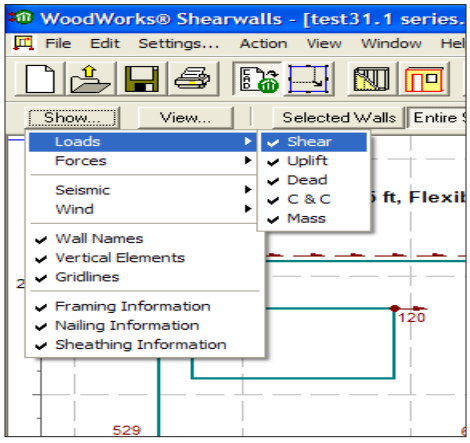
Data bars are located at the top of each window or "view". They allow users to change building levels, to turn data on and off in the corresponding view, and provide for quicker selection of options and settings than by the Settings dialog or the main menu. The buttons vary for each **view**.



The data bar for each view is described below:

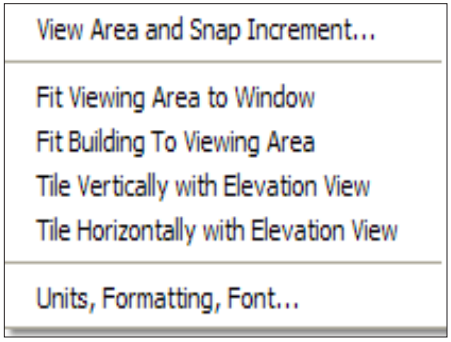
- **Plan View:** Show, View, Settings, Hold-downs, Log file, Current Level, Zoom In, Zoom Out, Undo, Redo, Getting Started
- **Elevation View:** Show, View, Selected Walls / Entire Shearline, Level
- **Results View:** Show, Settings, Preview, Go To Table

Show Button
This button is used to turn data on and off in the corresponding view.



View Button
This button is used to change viewing options.

Settings Button
This button is used to modify design settings.



Preview Button
This button is used in the Results view to create a print preview of the Design Results output report.

Log File
This button is used to view the results in a log file. For more information on the log file, see 2.13, Results.

Go To Table Button
This button is used in the ***Results*** view to select which table to view. For more information on accessing Results, see section 2.13.

Hold-Down Button
This button opens the hold-down database window where hold-downs can be created and designed. See section 2.16 for more details on hold down design

Other Data Bar Options

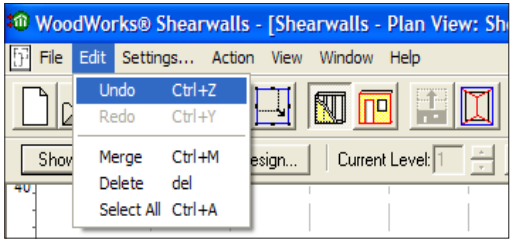
- **Current Level Button** – Controls the active level while in the Plan View. Only one level can be active while in this view.

- **Level Buttons** – Control the levels shown in the Elevation View and those for which output results are desired in the Results View.

- **Selected Walls / Entire Shearline Button** – Controls which walls along a shearline are shown in the Elevation View.

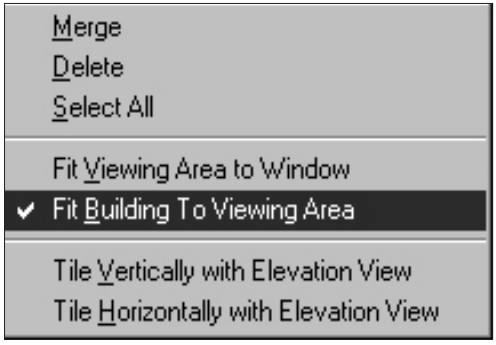
1.6 Menus

The main menu bar of the program contains all of the commands represented by the toolbar buttons. In addition, it contains several less commonly used commands, such as ***Print Preview***, ***Fit View To Window***, ***Select All Shearwalls***, ***Delete Shearwall***, and ***Merge Shearwalls***.



Context Menus

Context menus will pop up any time the user performs a right mouse-click. They contain some of the more commonly used menu and toolbar commands, and some extra shortcuts such as tiling the Plan View with the Elevation View.



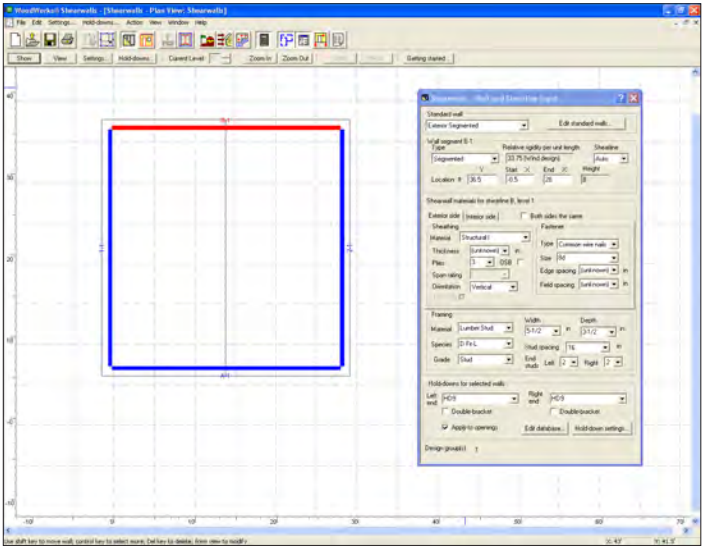
1.7 Views and Results

Shearwalls is divided into three main views in large windows that usually cover the whole viewing area, and several data input views that appear in a smaller window. The three main views are ***Plan View, Elevation View*** and ***Results View***. The data input window is known as the ***Form View,*** which is visible only when the Plan View is active, and allows for the input of structure, shearwall, opening and load data.

The data and viewing options for the Plan, Elevation and Results Views are controlled through the Data Bars or the Settings menu.

1. Plan View

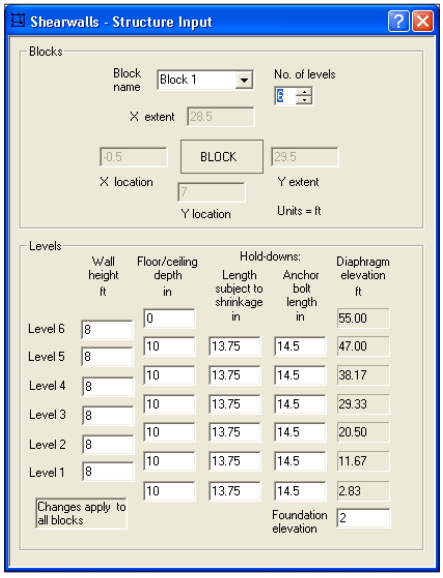
- This is the main working window for interactive graphical input of CAD drawings, structure outline, building levels, shearwalls, openings, and loads.
- This window is always present, but may be hidden by other windows. Pressing the Plan View button brings the Plan View window back into view.
- In Plan View, the user has the option of sizing the building to fit the current size of the window, or setting fixed extents for the window and using scroll bars, if necessary, to navigate around the building. The fixed extents are set in ***Settings/View***.
- Rulers appear along the sides of the plan view to show the scale, and the current mouse position is displayed in the ***Status Bar*** at the bottom right corner of the view.
- Gridlines are not named, or created by the user, as in Sizer. Instead, they are shown at regular multiples of the program's snap increment. Both the snap increment and the multiple are specified in the View settings.



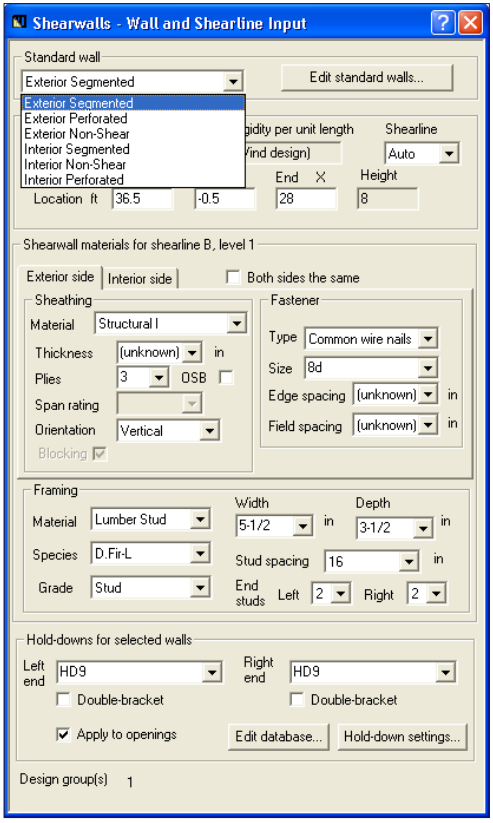
2. Form View

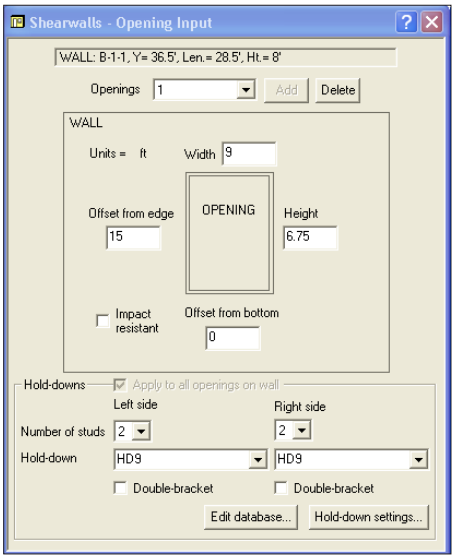
This view contains one of a set of forms for the input of data for building elements and loads. Each form contains data fields appropriate to the depressed Action button in the main toolbar:

- **Structure Input Form** – Available when the Structure action button is depressed, and is used to input the structure blocks, the number of levels for each block, and the wall heights and joist depths for each level.

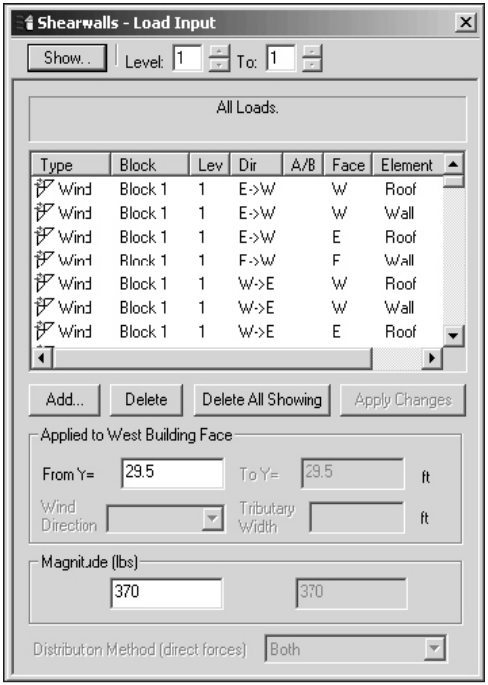


- **Wall and Shearline Input Form** – Available when the Walls action button is depressed. It is used to modify wall locations, and to input wall material data for the entire shearline (sheathing, fastening and framing details). It can also be used to create “Standard Walls” for future use.





- **Opening Input Form** – Available when the Opening action button is depressed. It is used to specify opening locations and dimensions.
- **Roof Input Form** – Available when the Roof Block action button is depressed. It is used to specify the roof geometry.
- **Generate Loads Form** – Available when the Generate Loads action button is depressed. It is used to generate wind and seismic loads.



- **Load Input Form** – Available when the Load action button is depressed. It is used to add and modify seismic, wind shear, C&C (suction), building mass, dead and uplift loads to wall lines and building faces.

The Form View displays the data for whatever object the user has selected in Plan View. (An item is selected by clicking on it with the mouse to highlight it in red).

When visible, the Form View remains on top of all other windows, so that the user can see the building elements and their data simultaneously.

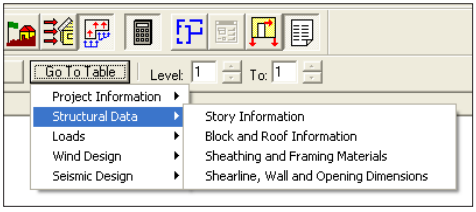
Pressing the Form View button makes the form disappear or reappear.

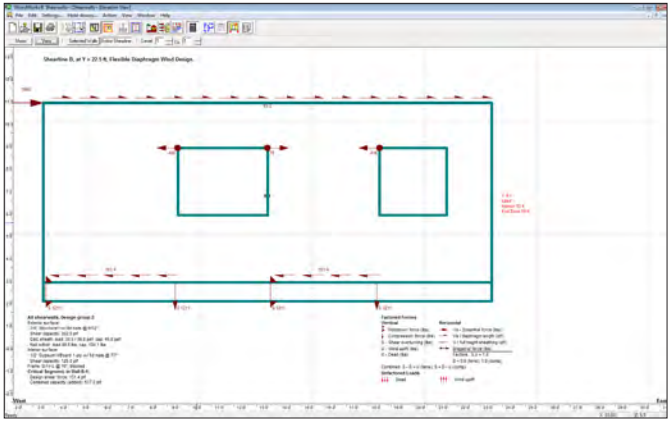
3. Elevation View

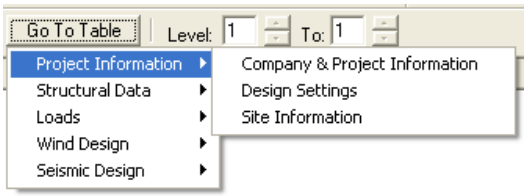
- Elevation view shows a drawing in elevation of the shearwall or walls selected in plan view.
- It will only show multiple walls if they are along the same shearline (a line of shearwall segments subject to the same loads).
- It displays either the walls selected in Plan View or all walls on the selected shearline.
- It will show all walls on the shearline for any range of building levels.
- It displays the dimensions of building elements; openings; applied shears, dead and uplift loads; and output data such as shear resistance; hold-down forces; dragstrut forces; nailing patterns required; and sheathing thickness.
- Pressing the ***Elevation View*** button creates the window if it does not already exist, and brings it into view.
- It is useful to tile this window with Plan View, where you can select the shearwall you wish to appear in Elevation View.

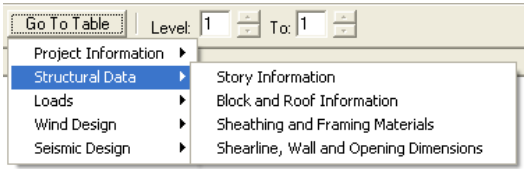
4. Results View

Results view shows a detailed design summary organized into the five main sections: Project Information; Structural Data; Loads; and Design (Wind, Seismic). These five sections can be accessed quickly once the Design Results is viewed by using the "***Go to table***" button in the ***Data bar***. Each of these contains several tables, described below.



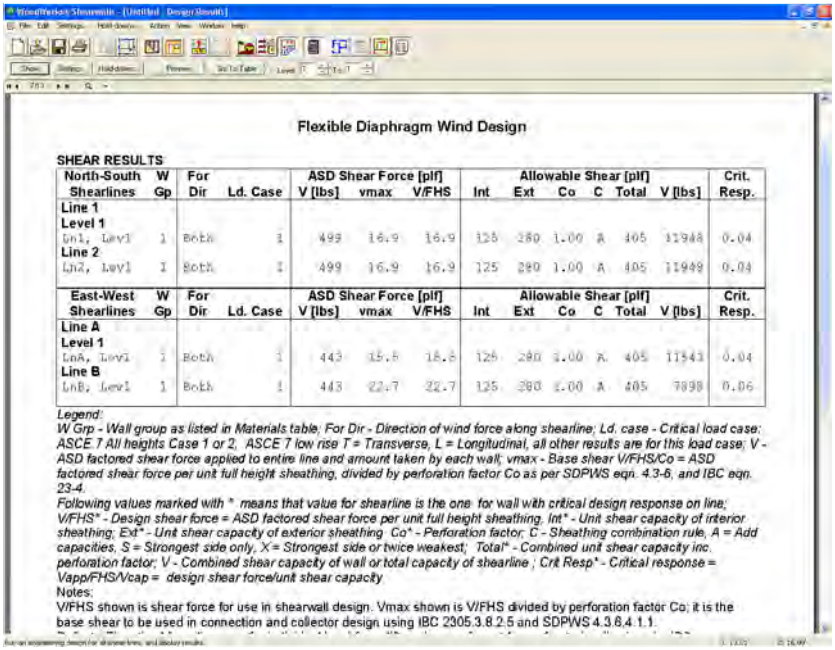






- **Project Information**: The ***Company Information, Project Description*** and ***Design Settings*** entered by the user in the ***Settings*** dialog are shown in the first two tables of this section. Data entered in the ***Load Site Information*** form is presented in the third table.

- **Structural Data**: The geometry of the building model is summarized in the ***Story Information*** and ***Block and Roof Information*** tables. The program creates Design Groups of walls with identical materials, and lists these in the two separate tables: ***Sheathing Materials by Wall Group;*** and ***Framing Materials by Wall Group***. These two tables, in conjunction with the designed ***Wall groups*** shown in the Shear Results tables, summarize



Flexible Diaphragm Wind Design

SHEAR RESULTS

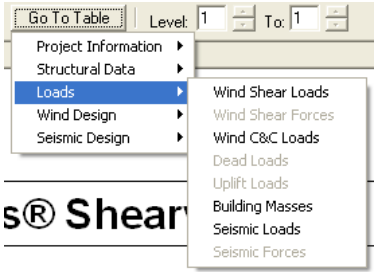
North-South Shearlines	W Gp	For Dir	Ld. Case	ASD Shear Force [plf] V [lbs]	vmax	V/FHS	Allowable Shear [plf] Int	Ext	Co	C	Total	V [lbs]	Crit. Resp.
Line 1													
Level 1													
Ln1, Lev1	1	Both	1	499	16.9	16.9	125	280	1.00	A	405	11948	0.04
Line 2													
Ln2, Lev1	1	Both	1	499	16.9	16.9	125	280	1.00	A	405	11949	0.04

East-West Shearlines	W Gp	For Dir	Ld. Case	ASD Shear Force [plf] V [lbs]	vmax	V/FHS	Allowable Shear [plf] Int	Ext	Co	C	Total	V [lbs]	Crit. Resp.
Line A													
Level 1													
LnA, Lev1	1	Both	1	443	15.5	15.5	125	280	1.00	A	405	11941	0.04
Line B													
LnB, Lev1	1	Both	1	443	22.7	22.7	125	280	1.00	A	405	7898	0.06

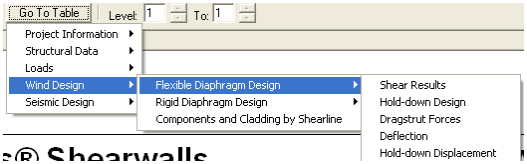
Legend:
W Grp - Wall group as listed in Materials table; For Dir - Direction of wind force along shearline; Ld. case - Critical load case: ASCE 7 All heights Case 1 or 2, ASCE 7 low rise T = Transverse, L = Longitudinal, all other results are for this load case; V - ASD factored shear force applied to entire line and amount taken by each wall; vmax - Base shear V/FHS/Co = ASD factored shear force per unit full height sheathing, divided by perforation factor Co as per SDPWS eqn. 4.3-6, and IBC eqn. 23-4.
*Following values marked with * means that value for shearline is the one for wall with critical design response on line; V/FHS* - Design shear force = ASD factored shear force per unit full height sheathing, Int* - Unit shear capacity of interior sheathing; Ext* - Unit shear capacity of exterior sheathing Co* - Perforation factor; C - Sheathing combination rule, A = Add capacities, S = Strongest side only, X = Strongest side or twice weakest; Total* - Combined unit shear capacity inc. perforation factor; V - Combined shear capacity of wall or total capacity of shearline ; Crit Resp* - Critical response = Vapp/FHS/Vcap = design shear force/unit shear capacity*
Notes:
V/FHS shown is shear force for use in shearwall design. Vmax shown is V/FHS divided by perforation factor Co; it is the base shear to be used in connection and collector design using IBC 2305.3.8.2.5 and SDPWS 4.3.6.4.1.1.

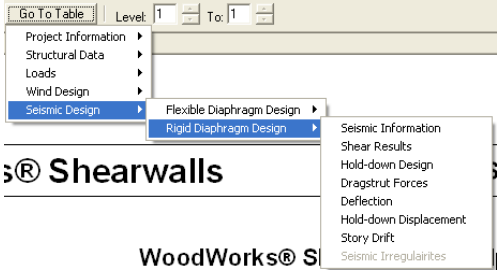
the shearwall construction requirements for the entire building. The dimensions and properties of individual walls, including the group to which they belong, are listed under ***Shearline, Wall*** and ***Opening Dimensions***.

- **Loads:** Dead loads, seismic loads and wind loads input by the user or generated by the software are listed in separate tables.



- **Design:** The ***Wind*** and ***Seismic Design*** Sections each include both flexible and rigid diaphragm design results. There are separate tables for shear results, hold-down design, dragstrut forces, shearwall deflection, and hold-down displacement. The ***Wind Design*** sections includes an additional table, Components and Cladding by Shearline, that compares the applied C&C wind suction load to the sheathing strength and fastener capacity in withdrawal. Only the ***Seismic design*** tables include story drift.
- The user can tailor the output reports using the **Show** button and can navigate from table to table with the **Go To Table** button.
- Design notes appear at the end of the wall materials table, and warnings are displayed where failed designs occur.
- The Results View window is created and displayed whenever a design is first performed.
- Pressing the **Results View** button brings this window into view for the most recent design run.



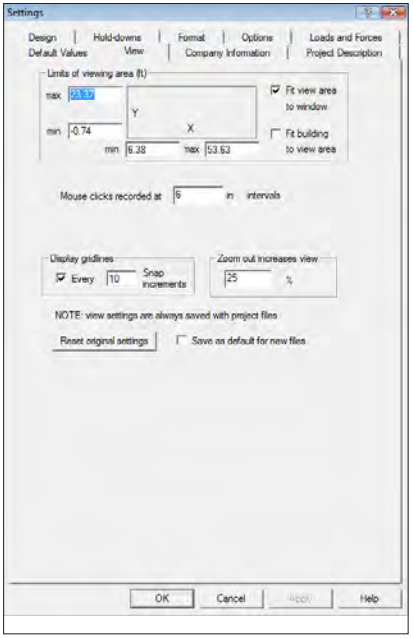


1.8 Settings

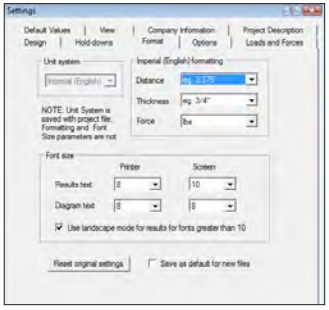
Shearwalls includes a comprehensive Settings dialog box that is accessed by pressing the ***Settings*** button in the data bar. It allows the user to control graphical interface options such as the units format, font sizes, view area and snap increments; to specify design settings; and to filter what will be shown in the Plan, Elevation and Results Views.

The Settings menu options are organized into seven tabs:

- **Design** – Controls the engineering design options, including the wind design method to be followed.
- **Default Values** – Controls the default materials, conditions, and dimensions used to generate the building and the loading model.
- **View** – Controls the viewing area limits, the snap increment and the gridline spacing.



- **Format** – Controls the unit format and the font size for the screen and printer output.
- **Options** – Control the text to be shown in the Plan and Elevation Views, and the tables included in Results View.
- **Loads and Forces** – Controls the loads and forces to be shown in the Plan, Elevation and Results Views.
- **Company Information** – Controls the input of company information to appear in the Design Results output.
- **Project Description** – Controls the input of the project description to appear in the Design Results output.



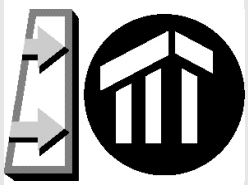
- **Hold-Downs** – Controls the hold-down offset and displacement options.

The group of settings on each tab can be saved as the default settings used by new files. The settings that came with the program can also be restored.

Refer to section B.4 for more in-depth information on Shearwalls Settings.

Designing for Wind & Seismic Forces

B.2



2.1 Getting Started

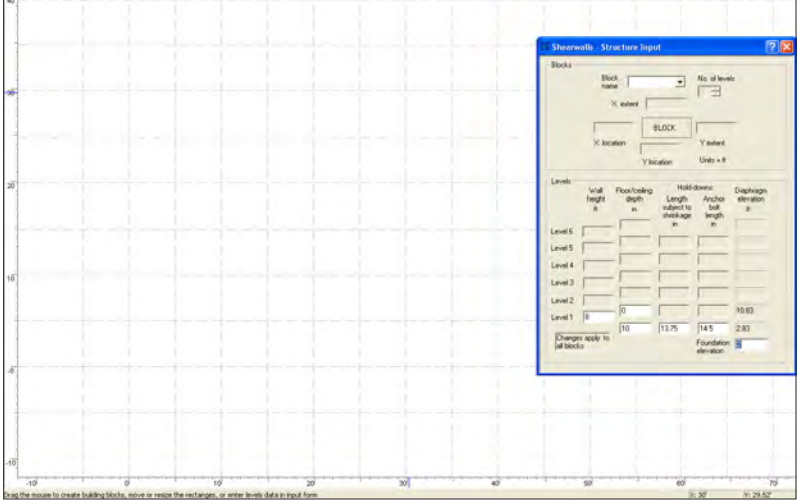
General

There are four main steps to designing the shearwalls in a structure with Shearwalls:

1. Define the building geometry by placing building blocks, specifying the number of levels and their elevations, and graphically modifying and drawing walls, using the mouse.
2. Using the Form View, provide details for each of the walls, for their openings, and define the roof geometry. Leave those items you wish the program to design as "Unknown".
3. Apply wind and seismic loads either generated automatically by the program or specified manually by you. Loads can be modified or regenerated at any time.
4. Run the design to determine the unknown values, calculate the forces and capacities, then display diagrams and the design results summary for all of the shearwalls.

Starting Off

Starting the Shearwalls program, or pressing the **File New** button, places you in ***Plan View***, with the ***Structure Action*** button depressed. At this point you may start drawing your main building blocks to define the building footprint, or "Structure Outline", or you may import a CAD drawing first.



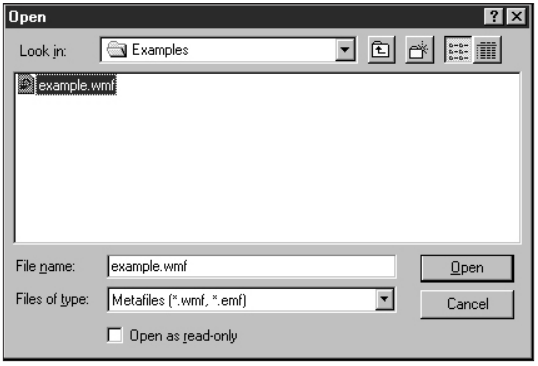
2.2 Importing a CAD File

Shearwalls has the capability of importing Windows metafiles (extension .wmf), independently for each floor. This is a file format that is an export option in CAD programs such as AutoCAD®.

Important Note: You must use a white or light background with black or dark lines and text for your CAD file. Dark backgrounds are incompatible with the Shearwalls screen graphics.

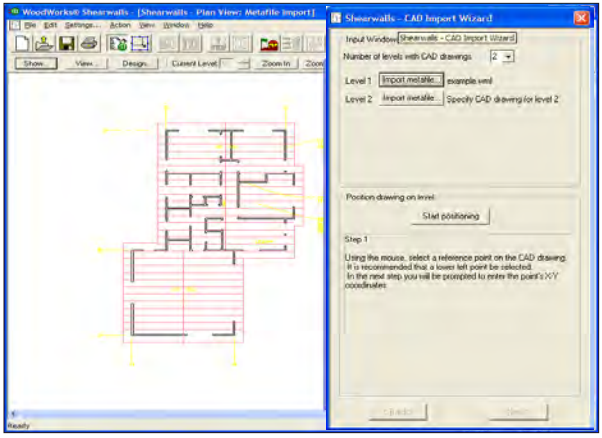
The ***Import*** function opens these files and displays the drawing in Plan View for the user to trace over and to recreate the structure. Afterwards, the user can view or hide the original drawing at any time by pressing the Import button.

1. Click the ***Import CAD Drawing*** button on the toolbar.
2. Locate your CAD files using the dialog box and ***Open*** it.



3. The CAD drawing will be displayed in Plan View, and the rulers will temporarily disappear.

The Form View will display a form called ***CAD Import Wizard***. Follow the instructions to position and scale the CAD drawing in the coordinate system of Plan View. Once this is done, rulers will appear on both axes showing the new scale. Use the Zoom buttons, and the scroll bars to resize or reposition the drawing on the screen. The limits of the viewing area and the snap increments can be set from the **View** tab of the **Settings** dialog (select **Setting/View**).

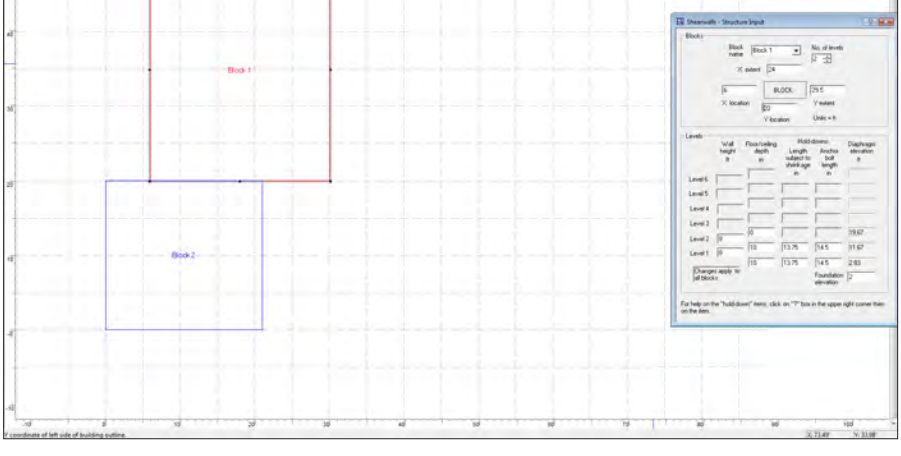


Note: Once you have positioned the last level of the structure and exited the CAD Import Wizard, it is not possible to re-enter the Wizard to change a CAD Import file. Choose your imported files carefully!

2.3 Creating Building Blocks

Footprint

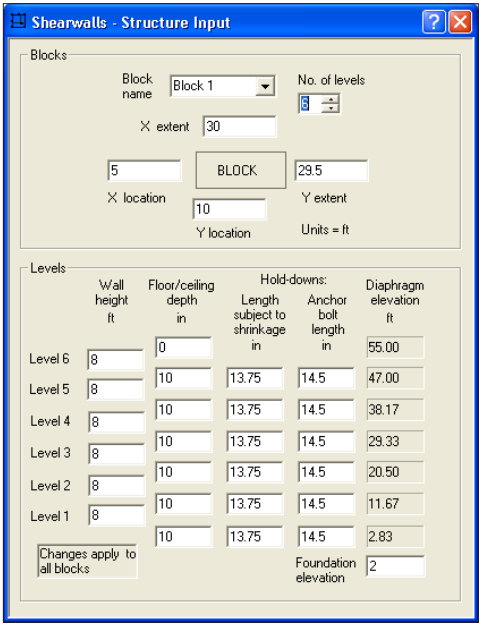
1. When the program opens, a new file is started, or you have just imported a CAD drawing, you will be in **Plan View** with the ***Structure*** button selected.
2. Click the mouse at the point where you want one of the corners of your first building block to be.
3. Drag the cursor to another point and release the mouse button to create your first block.
4. Resize or reposition the block by highlighting it by clicking on a handle and dragging it, or by changing the location or extents that appear in the **Form View** for **Structure Input**.
5. Block names can be specified by editing the **Block Name** field. The number of levels for a block, hold-down lengths and general level information can also be specified at this point.
6. Additional blocks are created by following steps 2 to 5 above and by ensuring that new blocks abut or overlap an existing block. New blocks being created cannot abut or overlap more than one of the existing blocks.
7. When exiting **Structure Action**, walls will be placed automatically around the edges of the intersecting blocks. You will not be able to move, resize or reposition the blocks while in Walls action, but the blocks will automatically resize to follow movements of the exterior walls.



2.4 Creating Storeys

The building levels are defined in the ***Structure Input Form*** before proceeding to the rest of the design sequence. **The number of levels cannot be modified once the Extend Walls button or the Roof Blocks button has been pressed, but the story data can be changed at any time.**

1. While in the Structure action in Plan View, the **Structure Input** Form will appear. (Click on the Form button if this Form is not showing).
2. Highlight one of the building blocks and in the **Levels** portion of the Form, select the number of levels for your block by clicking on the arrows or by entering a value. The maximum number of levels is **six**.
3. Specify the number of levels for all remaining blocks.
4. The program will automatically enter a **2 ft.** US version imperial / **1 meter** (Cdn, metric) foundation elevation, and the default wall height and floor depth values (as defined in Settings/Design) for each level.
5. Edit the foundation elevation, the wall heights and floor depths for each level. This information applies to all blocks, so that the floor elevations are the same for each block. The blocks can have different numbers of levels.



Extending Walls

The last step in creating levels is to extend the first floor upwards once the elevations have been specified and walls and openings have been created for the first level.

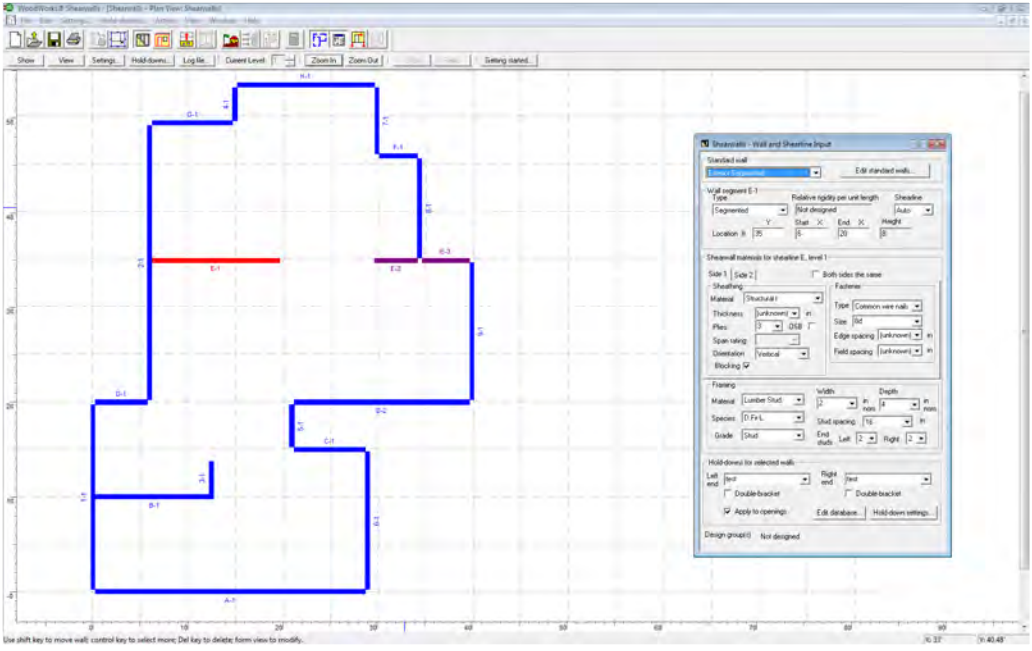
1. When in Walls action or Openings action, click on the **Extend Walls** button. All exterior walls defined for the first level will be copied to all levels above for all blocks up to the maximum number of levels specified for each block.
2. Note that the user may choose to extend the levels after modeling the first floor for only those elements that are common to all levels.
3. The Extend Walls button can only be applied once.

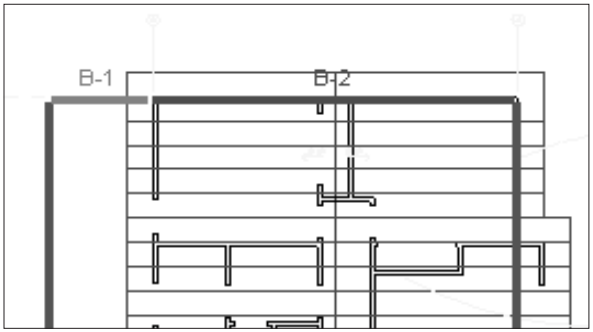
2.5 Working with Walls

Clicking on the ***Walls*** button allows you to manipulate all the walls on the screen, and displays the ***Shearwalls*** data form in the ***Form View***.

1. **Selecting Walls**
- You can select a wall or wall segment by clicking on it. It will then appear red.
- The data for this wall will appear in the Form View, and you can perform any of the graphical or text editing procedures on the wall that are described below.
- You can select multiple walls by keeping the CTRL key depressed while you select walls. Those data common to the selected wall will appear in the Form View, and certain editing actions will be available.

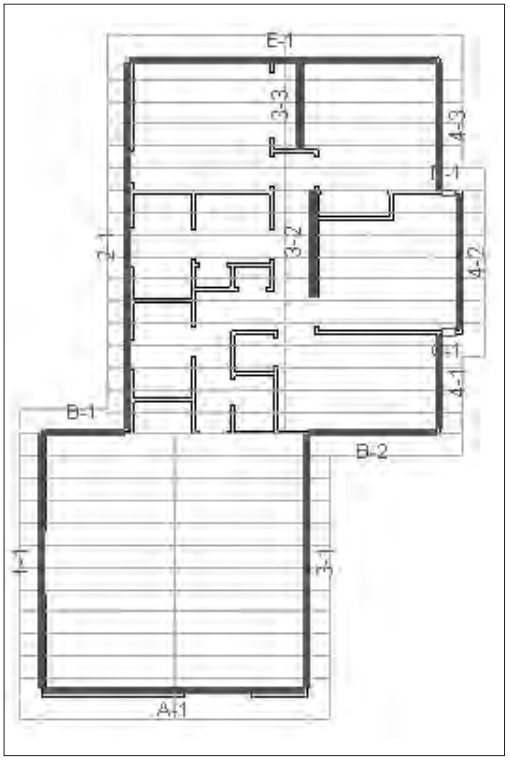
2. Creating Walls
- New interior walls can be created by clicking the start point of the wall, dragging the mouse to the endpoint, and then releasing the mouse button.
- It is not possible to create new walls outside the footprint of exterior walls that were created in the Structure Outline View.
- The wall will have the same shearwall type and be composed of the same materials as the currently selected “Standard Wall” in the Wall Form View.





3. Subdividing Walls

- Wall segments can be created from existing walls in order to give them different properties, or to offset portions of a wall from the rest of the wall.
- Click on a point anywhere along an existing wall, drag the mouse to the end of the segment and then release the mouse. The wall will then appear broken into 2 or 3 segments, each with its own name.
- The new wall segment will have the same properties as the wall it was created from.



4. Moving Walls

- Entire walls and wall segments can be moved perpendicular to themselves.
- Hold the SHIFT key, then click the mouse anywhere along the wall to be moved and small arrows will appear around the line.
- Drag the mouse to the new position of the wall, then release. Note that walls cannot be repositioned past other walls.
- When moving a wall segment, two new walls will appear perpendicular to the wall being moved, to maintain connectivity with its previous position.
- In this way, extensions to the building can be made quickly.
- When moving an entire wall, the connecting perpendicular walls will be stretched or shrunk to maintain connectivity.

5. Resizing Walls

- Press the SHIFT key, then click on the endpoint of a wall, or where the ends of several walls meet; these walls will be selected and appear red.
- Small arrows will appear indicating the direction that you must move the mouse to resize the walls.
- Drag the mouse in that direction, and release it where you want the new endpoint to be.
- Both walls that are selected in one direction will be resized. Other exterior walls may be moved to maintain the integrity and connectivity of the structure.

6. Merging Walls

- You may wish to undo a subdivision of a wall, or to combine a wall segment with an adjacent one.
- With the CTRL key depressed, select the walls to be combined.
- Select the ***Edit/Merge*** command from the main menu or from the right mouse button context menu.
- The combined wall will remain selected. It will have the properties of the first wall selected.

7. Deleting Walls

- Select any number of walls, and press the delete key or select the ***Edit/Delete*** menu item.
- Interior walls will disappear entirely. Exterior walls will be merged with adjacent walls.

Shearlines

A shearline consists of one or more parallel shearwalls that act as one to resist an applied shear load. A shearline may contain gaps or non-shearwall components. A wall can be designated as a non-shearwall should the designer wish to neglect its contribution to a shearline. The walls may be on different wall lines and considered as part of the same shearline as long as they are separated by no more than the maximum shearwall plan offset. The maximum shearwall offset is defined in the **Settings Design** tab, and is defaulted to the following values:

- U.S. – The default offset is **6 inches.**
- Canada – The default offset is **150 mm.**

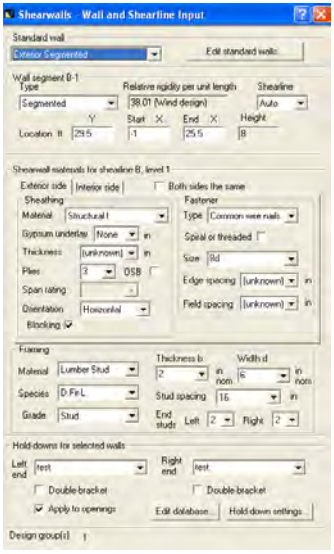
Note: A warning is generated in the output if the offset is greater than zero for the Canadian version.

A bandwidth approach is used to determine which walls belong to a shearline, where the bandwidth is equal to the maximum shearwall offset. For example, N-S running shearlines are created from left to right on the plan layout. The first N-S wall encountered becomes the left limit of the first shearline's bandwidth. Any wall found within the bandwidth is considered to be part of the same shearline.

The next N-S shearline starts at the first wall found beyond the previous shearline's bandwidth.

Shearlines are automatically generated by the Shearwalls program. They are created or deleted as walls are created and moved about. If a wall can belong to more than one shearline, it can be redesignated using the Wall Input form.

Shearlines are not drawn in any of the views but when you select a wall to load in Plan View all walls on the shearline will be highlighted in purple. The text output is generated on a shearline by shearline basis.



Changing Wall Properties

If the Form View is not visible, press the **Form View** button on the main toolbar. In Plan View, select a wall by clicking on it with the mouse. It will appear in red. Going back to the Form View, you can then change any of the properties of the wall as follows:

1. **Wall Type (U.S. Version)**

- Specify the type of wall by selecting from the drop-down menu for Shearwall Type. It can be a Non-Shearwall (not designed as a shearwall) or a Shearwall. A Shearwall can be identified as a Segmented or Perforated wall.

2. **Hold-down Configuration (Canada Only)**

- Specify the location of hold-down connections by choosing one of the following options from the pull-down list:

Non-shearwall
The wall is not designed as a shearwall.
All segments
Hold-downs are placed at each end of the wall and on either side of every opening, except where there is no net overturning.

Ends of shearwall and where required
Hold-downs are placed at each end of the wall, as well as at the end of any shearwall segment where required (based on material selections and anchorage restriction settings).

Ends of shearline and where required
Hold-downs are placed at each end of the shearline, as well as at the end of any shearwall segment where required (based on material selections and anchorage restriction settings).

Where required only
Hold-downs are placed only where they are required (based on material selections and anchorage restriction settings).

Note: There is a trade-off between the use of hold-downs and the materials required for shearwall design. For example, designing without hold-downs may require thicker panels and/or a tighter nail spacing to achieve the required resistance.

Use the **Hold-down Configuration** pull-down in conjunction with the **Anchorage Restriction Settings** (from the Design Tab of the Settings menu) to specify which should take precedence in the design: use of hold-downs or the materials selected

for shearwall design. Refer to the on-line help for a detailed description.

3. Standard Walls

- If you select from the list of "Standard Walls" while a wall is selected, all of the properties for the standard wall are transferred to the selected wall.
- If you change a wall so that it becomes identical to a Standard Wall, that Standard Wall name will appear. If you make a change so that it is no longer the same as a Standard Wall, a blank space will appear.
- Press the "Edit Standard Walls…" button in order to add, delete or modify standard walls. Existing walls will not be affected by changes made to standard walls other than to cause the Standard Wall field to be cleared.

4. Wall Dimensions

You can:

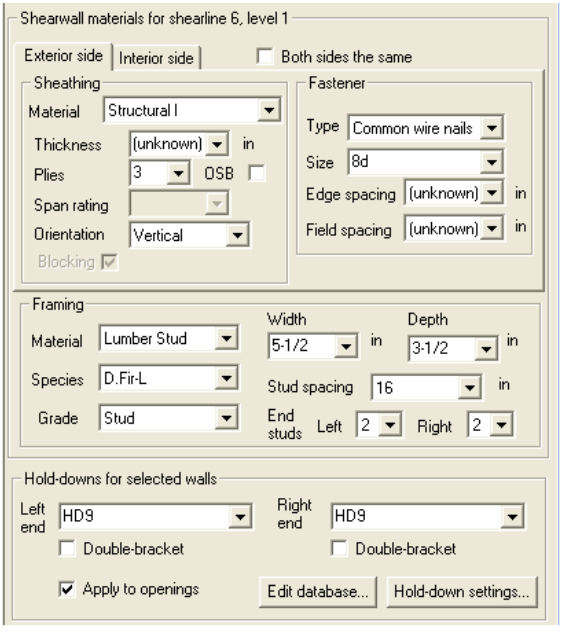
- Change the location of the wall. This has the same effect as moving the wall graphically, as described under 'Moving Walls', earlier in this section.
- Change the start or end of the wall, with the same impact as resizing the wall graphically.

5. Wall Materials

To change the material properties of the wall:

- First you must specify which sheathing surface the changes apply to: exterior, interior, or both sides. (Selecting "Exterior Only" indicates that there is no interior sheathing.)

- Next, specify the type of sheathing material. This causes a new list of choices for sheathing thickness to appear. These thicknesses are the ones that yield differences in design strengths (U.S. only). If you are using a slightly different thickness, just select the next available smaller thickness from the list. You may also specify "*Unknown*" and the program will design a thickness for you. You can also specify the orientation of application.
- Now select Fastener, Type, Size, and the Nail Spacing for the edges and for the interior of the sheathing panel. Choosing "*Unknown*" for edge spacing allows Shearwalls to determine this value during the design process.
- Finally, specify the framing details, material, species and spacing of the lumber studs, and whether there is blocking between them. Stud Spacing may be "Unknown"



6. Multiple Walls

- To change the materials or wall type of several walls at once, select the walls in Plan View using the **CTRL** key, or from the Edit menu or by right clicking, choose **Select All** to highlight all walls on the level.
- If not all of the selected walls share a certain property, that field on the input form will become blank.
- Any changes to the properties will now affect all the selected walls.

Hold-Downs for Selected Walls
Select the type of hold-down for the selected wall. You can also access the hold-down database to create new hold-downs to be used in the design and the hold-down settings window.

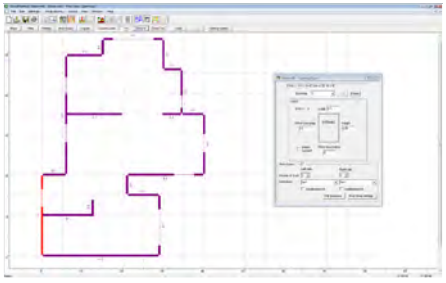
Selecting ***Apply to openings*** will specify these same hold downs for each window and door opening in the selected wall(s). Selecting a ***Double bracket*** means that there is a hold down above and below the bottom of the wall, typical for floors above the first level, and will double the hold down elongation/displacement for the purpose of calculating deflection.

2.6 Creating Openings

1. Click on the ***Openings*** button. The input form will display information about the openings (windows and doors) in the currently selected wall.

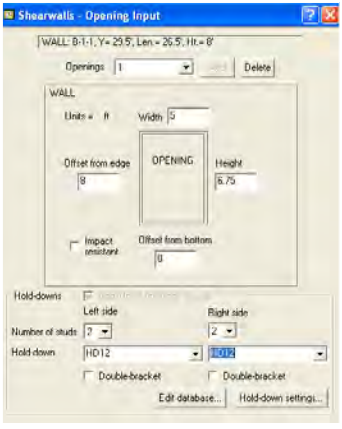
2. In ***Plan View,*** you can draw any number of openings in a selected wall by dragging a line from one point on a wall to another.

- The opening will appear as a thinner line than the wall.
- The opening will initially have a height and an offset from the bottom as specified in the ***Settings/Design***.



3. To change the location or the dimensions of an opening:

- Select the opening number in the drop-down box at the top of the form. The current dimensions will appear in the form.
- Type over the dimension you wish to change and press **Enter**.
- Note that openings cannot be altered graphically after they have been created: they must be resized or relocated using the Form View.



4. It is possible to create a new opening without using the Plan View.

- Select ***"new opening"*** in the **Openings** drop-down list, enter the dimensions and location of the opening, and then press the ***Add*** button.
- You can add a succession of similarly sized openings in this way.

5. Deleting openings:

- To delete an opening, click on a wall, select an opening from the **Openings** drop-down list, then press the ***Delete*** button.

6. Hold downs are specified for each side of each window and door opening. If the *Apply to all openings on wall* is selected, the choice of hold down will apply to all the openings along the selected wall. Unchecked, unique hold downs can be specified for each opening. By default, this option is disabled, and checked, so that a wall with multiple openings have consistent hold downs. To enable the checkbox, the "*Apply to openings*" in the ***Wall and Shearline Input*** form needs to be unchecked. To reset the hold downs entered in the ***Openings Input*** form to match those entered for the wall, recheck the "*Apply to openings*" or choose a hold down for the left and right end of the selected walls in the ***Wall and Shearline Input*** form. The ***left end*** in the *wall and shearline input* corresponds to the "***left side***" in the ***Opening Input*** form. It would be more typical to specify a consistent hold down for the entire wall system, perhaps even the entire building.

2.7 Creating Roofs

Roof Action becomes available for single story buildings when the Wall Action button is pressed and walls are created, or for multi-story buildings when the *Extend Walls Upwards* button is pressed. You must press on roof action before proceeding to create loads.

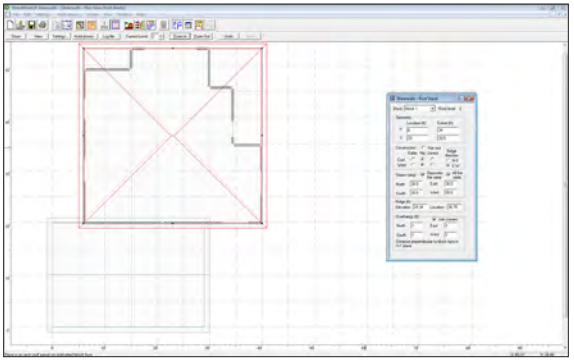
Click on the ***Roof Blocks*** button to specify the building roof geometry. A Roof Block is created for each structure Block. Each Structure Block will be assigned an initial roof geometry based on the construction type, slope and overhang settings in ***Settings/ Default Values.***

After the Roof Blocks button is pressed for the first time, the exterior walls remain associated with the blocks. If these walls are repositioned, the roof block is resized to match their furthest extent. However, the opposite is not true, It is also possible to create roof blocks that are not associated with any walls by dragging them on the screen while in Roof View.

Similarly, if the exterior walls associated with a particular block are repositioned, the roof block (and structure block) will adjust itself to match the furthest extents of the exterior walls. The opposite is not true.

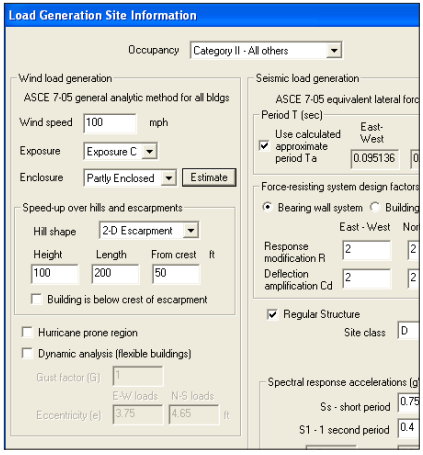
Adjusting Roof Blocks
The Roof Input Form controls the editing of roof blocks to define the roof blocks location and geometry.

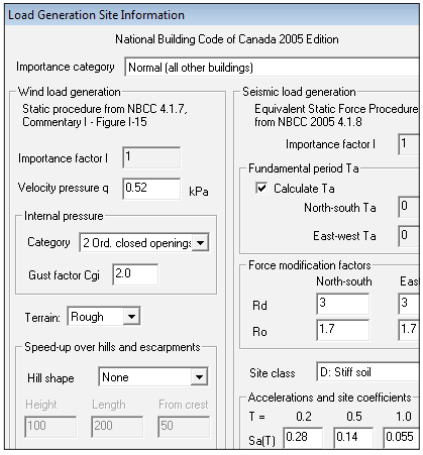
1. Select a Roof Block by clicking on it in plan view, or by choosing it in the Block pull-down in the Roof Input Form. Selected blocks are highlighted in red.
2. Resize or reposition the block by clicking on a handle and dragging it, or by changing the location or extents that appear in the Form View for Roof Input.
3. Define the roof construction, ridge direction, roof slopes, ridge elevation, plan location and size of overhangs by editing the appropriate fields in Form View.
4. To delete a roof block, select a block and then press the Delete button.



2.8 Specifying Site Information

The ***Building Site*** information pertains to characteristics of the building such as period, enclosure, exposure, and occupancy; and climatologic, topographic, seismologic and soil characteristics of the site. The site information is used for the automatic generation of wind and seismic loads.





The initial building site information is based on the settings in **Settings/Default Values** and **Settings/Design.**

The ***Building Site*** dialog box can be accessed while in any of the Views and Actions, and after entering the data you return to the previous state.

Occupancy / Importance Category (U.S. / Cdn)
The building *Occupancy* (US) / *Importance category* (Canada) is selected based on the nomenclature used in the building code selected for design.

Wind & Seismic Load Generation
Depending on the design method standard selected for wind generation (refer to ***Settings***), the input fields for wind load generation will vary and certain fields may become inactive. Refer back to section 1.2 ***Design Methods*** for more information

Refer to the ***Help*** for more information on *Terrain* (Cdn)/ *Speed up over hills* (US), *Hurricane prone regions* (US), and *Dynamic Analysis (flexible buildings)* (US, wind only), as well as other specific input criteria for wind and seismic generation.

NOTE: the Dynamic Analysis (flexible buildings) allows a manually calculated entry of Gust factor suitable for flexible buildings. However, it would be highly unusual that any wood structure, even midrise, to be considered a "flexible" building. "Flexible" buildings are typically defined as being slender with a fundamental natural frequency of less than 1Hz (that is, a period of greater than 1 second). Almost all wood structures are considered "rigid", with a period of less than 1 second. The fundamental (Cdn) / approximate (US) period Ta shown in the Load Generation Site Information input will help determine if the designed building can be considered "Rigid" or "Flexible". Note also, that the determination of a "flexible" or "rigid" building is not related to the distribution of loads based on a "flexible" or "rigid" diaphragm.

2.9 Automatically Generating Loads

Click on the ***Generate Loads*** button to generate wind and seismic loads. Wind or seismic loads can be generated separately or at the same time by selecting the appropriate type of loads to generate. Loads can be generated on the entire building, on a range of levels or on a single level by choosing the appropriate levels in the *Generate Loads* form.

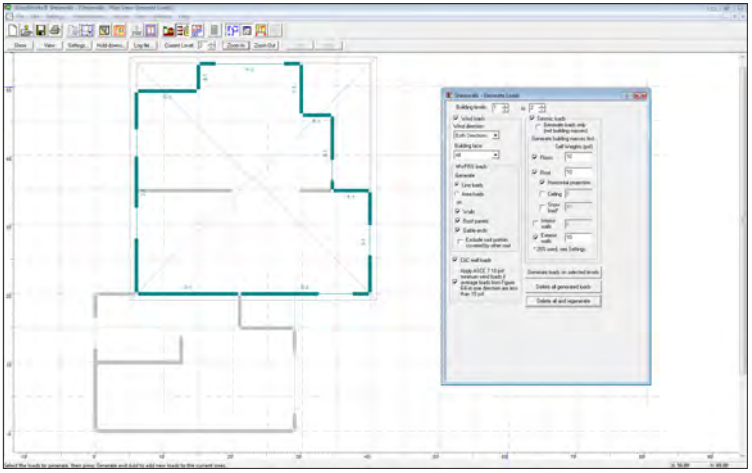
Wind Loads
Use the wind direction pull-down to generate wind loads for all directions at once or for a single direction only. Similarly, use the Building Face pull-down to generate loads on different building faces.

MWFRS (Main Wind Force Resisting System) loads can be generated as either line loads or area loads. Line loads is the default, but both produce the same results.

Loads can be generated individually for walls, roofs and gable ends. Gable ends are considered as the portion of the wall at the gable end that is above the eave. The portion of the wall below the eave is considered in the Walls selection. C&C (components & cladding) wall loads can also be generated.

In the US, WoodWorks® allows you to select the option of generating a 10 psf minimum wind pressure. In the All-heights method, the 10 psf minimum option is selected automatically and the generated pressures will be the higher of the "wind speed" generated loads and the 10 psf pressure. In the Low-Rise method, the 10 psf minimum pressure is not selected as default and must be analyzed separately from the "wind speed" generated cases by selecting and deselecting the 10 psf minimum option.

For Rigid Diaphragm Analysis using the All-heights method, the ***Settings*** form allows you to select between two separate cases: full wind pressures without torsion and reduced wind pressures with torsional effects. This analysis requires separate runs and must be compared to determine the worst case. Refer to Section B.4 for further information on Shearwalls Settings.



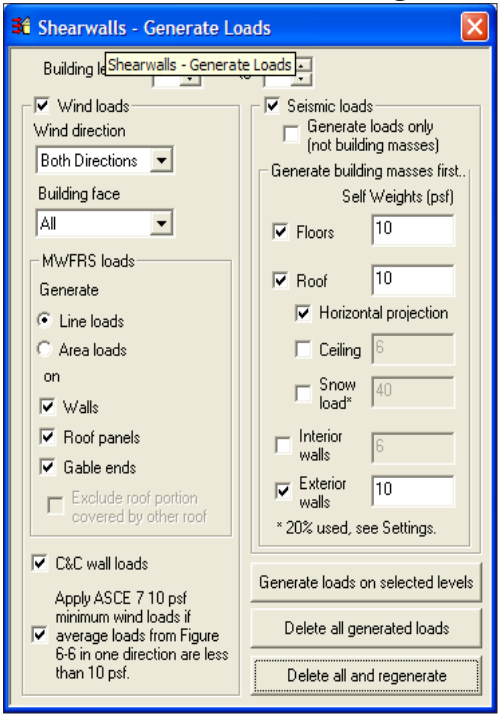
Seismic Loads
The loads generated correspond to building masses that are also generated by the program. Check the box for each building element - walls, roofs, floors and ceilings, that you want the masses to be generated for, and specify the area self weight of these elements. The self weight specified for the roof can be input as acting along a horizontal projection of the roof or along the surface of the roof.

The ***Generate Loads*** button allows you to use different self weights for different levels, by selecting each level in turn, changing the self-weight and generating the loads for that level.

The initial self weights used to generate seismic building masses are based on the values in ***Settings/Default Values.***

Refer to Section B.4 for further information on Shearwalls Settings.

Use wall self-weights to generate wall dead loads for Jhd calculations (Canadian Version)
Activate this checkbox to include the self-weight of walls to calculate wall dead loads that are considered to resist overturning. Note that the self-weight of floors, ceilings and the roof, entered in the Generate Loads form, are only used to generate building masses which are used to calculate seismic loads – they are not considered to resist overturning.

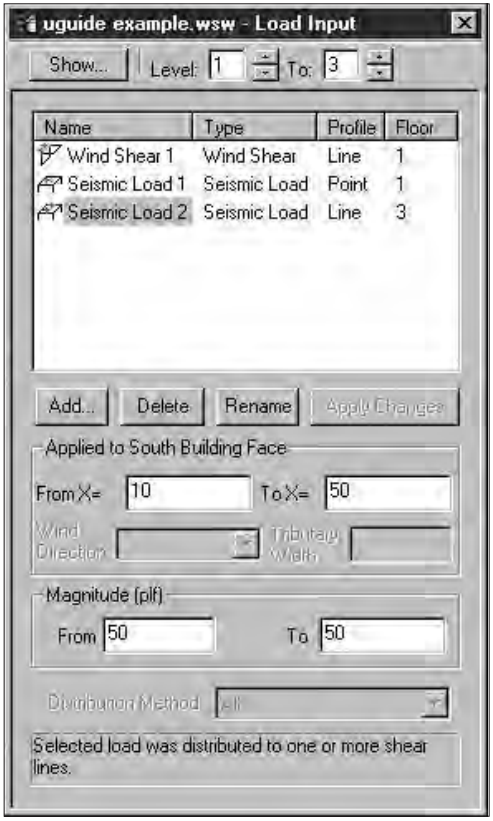


2.10 Manually Applying & Modifying Loads

Click on the **Loads and Forces** button to add a variety of load types and profiles to the structure, as described below in the *Load Input Form* section. Shearwalls has the flexibility to add loads to one or several levels at once, to individual walls or entire building faces.

Load Input Form

This form controls the adding, editing and deleting of loads and forces that can be applied to the structure.



The loads list can be controlled to only show certain categories of loads:

- ***Show… Button*** – Show loads according to their type (Seismic, Wind Shear, Wind Uplift, Dead, C & C).
- ***Level Control*** – Show loads according to the level on which they are applied. This can be done for one or a range of levels.
- ***Selected Building Face*** – Shows loads that are only applied to the selected building face in Plan View.

Load Types

Shearwalls accounts for the following load types:

- ***Seismic*** – Earthquake generated loads restricted by the lateral load resisting system.
- ***Wind Shear*** – External wind loads resisted by the lateral load resisting system.
- ***Wind C&C*** – Wind suction effects on the Components and Cladding for exterior walls only (roofs not included). This includes the bending of sheathing between studs and the withdrawal of nails fastening the sheathing to the wall studs.
- ***Wind Uplift*** – Overturning loads, especially wind uplift load transferred from the roof to supporting walls on the top story.

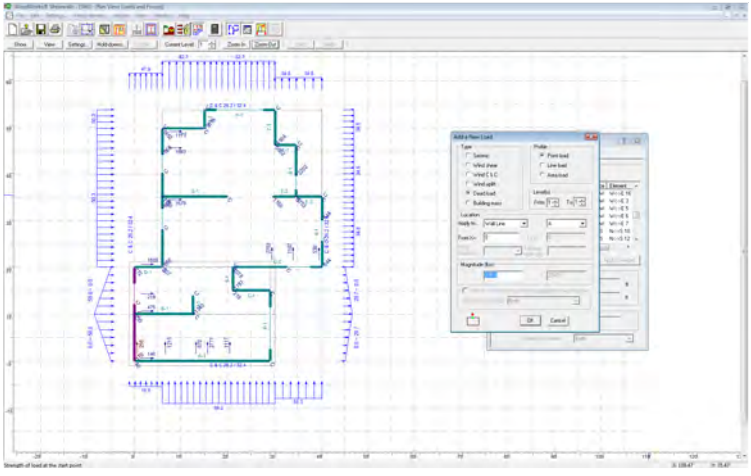
- ***Building Mass*** – Weight of a building attributable to the diaphragm that transfers loads to the specified level. This weight is used for automatic generation of seismic loads (see Section 2.9) only, and should include all material self-weights along with any roof snow loads or permanent dead loads required by the building code. Building Mass is not considered to resist overturning.

Load Profiles
A variety of load profiles can be applied to the structure, including point loads, line loads and area loads. Loads can be applied to one or more floor levels.

Load Location
Loads are applied to building faces, wall lines, or selected walls. The location and tributary area can be changed to create partial loads or overhanging loads. Wind shear loads can be applied as windward and/or leeward loads. Highlight a wall in Plan View to add corresponding loads.

Magnitudes
Load magnitudes represent the intensity of the load applied to the exterior of the structure. These loads are based on the code requirements for your jurisdiction.

Implement as a Force Applied Directly
Refer to section 2.11 under Direct Applied Shearline Forces for a description of this feature.



Working With Loads

1. **Adding Loads**
- Click on the **Loads Action** button to input loads through the ***Load Input Form***.
- In the Load Input Form select ***Add***… to add loads to the structure. The ***Add a New Load*** input dialog will appear.
- Select the load type, profile, location and floors to which the load applies.
- The magnitude is entered as a From and To value to facilitate trapezoidal and triangular line loads.
- Area loads require a tributary width.
- Click **OK** to add the new load to the list of loads in the Load Input Form.

2. Editing Loads
- Select the load to be edited from the *Load Input Form.*
- Edit the load location or magnitude information as required and click on the **Apply Changes** button. The load type or profile cannot be changed.

3. Deleting Loads
- Select the load to be deleted from the *Load Input Form.*
- Click on the **Delete** button.
- To delete all loads, click on **Delete all showing** button.

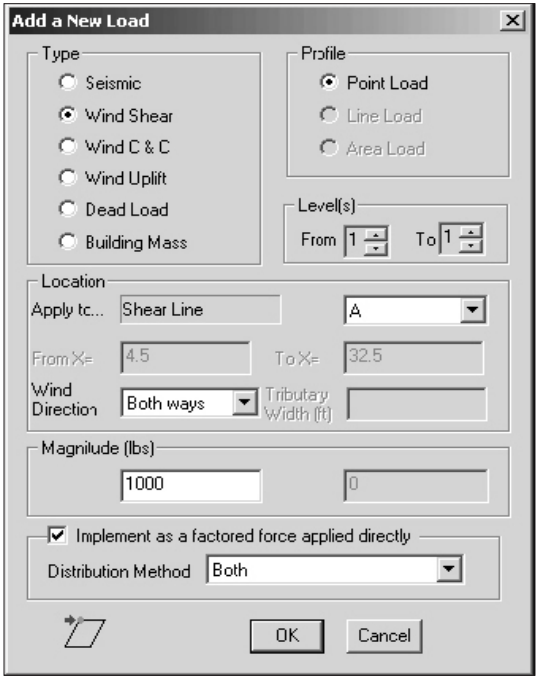
To edit or delete individual loads, select the wall in *Plan View* to which the loads are applied. The loads list in the Load Input Form will now display only those loads applied to the selected building face.

Note: After editing generated loads, you must perform a design before you regenerate the loads, otherwise your changes will be lost when the new loads are generated.

2.11 Generating Shearline Forces

For the Flexible diaphragm analysis method, Shearwalls automatically distributes loads to the shearlines each time a new load is added. It also adjusts the distribution of forces if changes are made to the building's walls.

For the Rigid diaphragm analysis method, loads will be distributed to the shearlines once the Design button is pressed after creating or adjusting loads or making changes to the building's walls. The rigid method considers the torsional resistance of walls both parallel and perpendicular to the applied load, therefore all walls in all directions must be loaded in order to perform a rigid diaphragm analysis.



Direct Applied Shearline Forces
As an advanced feature, Shearwalls allows forces to be directly applied to shearlines as a manual method of adjusting the load distribution.

A force magnitude can be entered as either a positive or negative value to facilitate redistribution of the loads by adding forces to some shearlines and subtracting forces from others (The load distribution method can be Flexible Diaphragm, Rigid Diaphragm or both. Most often, both methods will be selected).

These direct forces are added in the same manner as for new loads:

- Click on the **Loads** button.
- Select or highlight in Plan View the wall in the shearline that the force applies to.
- Click on the **Add…** button and select **Implement as a Force Applied Directly.**
- Specify the force direction in the case of wind, enter the magnitude and select the load distribution method to which the force applies.

Direct forces are not shown in the plan or elevation views as applied loads. Rather, the results of applying these loads are shown as adjusted shearline design forces in both the *Plan* and *Elevation* views. Direct forces appear in the results view.

There is a lot of additional information in ***Help*** pertaining to generation and distribution of loads, and the related engineering assumptions used in the software.

2.12 Viewing Loads, Forces & Building Masses

Applied loads and corresponding forces can be viewed in the Plan and Elevation views.

First click on the **Show Button** to select the loads/forces to be viewed. Viewing is made easier if wall names are turned off.

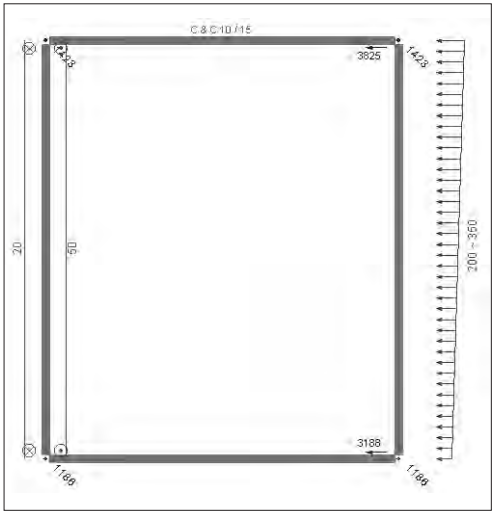
Forces in *Plan* and *Elevation* views can be shown based on either the *Flexible* or *Rigid* diaphragm analysis methods for the distribution of lateral loads to shearlines. A torsional analysis is performed for the rigid method.

> Note: Loads appearing are the sum of all loads of a similar type that have been applied to the building face at the current level.

Plan View

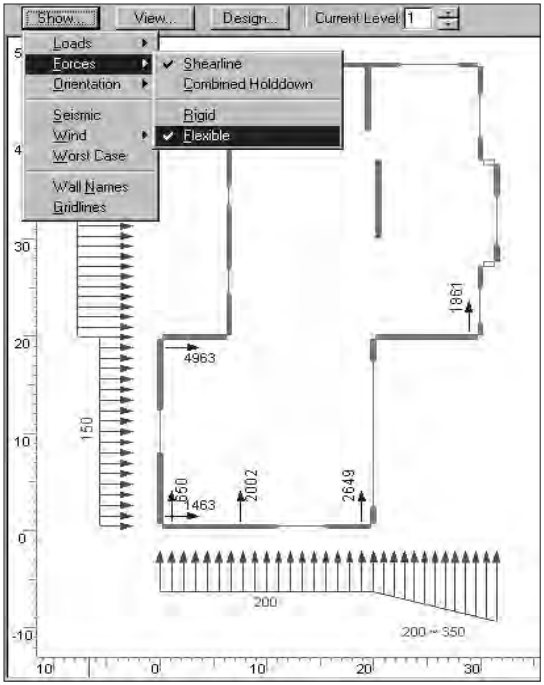
Both loads and forces can be viewed in the Loads and Forces view, while forces cannot be viewed in the Generate Loads view.

- A **seismic load** is distributed (point/ uniform/ trapezoidal) over the length of an exterior wall as horizontally applied double-headed arrows with specified magnitude. The corresponding horizontal shearline forces appear as double-headed arrows with calculated magnitude shown at mid-length of the affected shearlines. The calculated magnitudes of the hold-down forces are shown at the end of each wall segment.
- A *wind shear load* is distributed (point/ uniform/ trapezoidal) over the length of an exterior wall as horizontally applied single-headed arrows with specified magnitude. The corresponding horizontal shearline forces appear as single-headed arrows with calculated magnitude shown at one end of the affected shearlines. The calculated magnitudes of the hold-down forces are shown at the end of each wall segment.

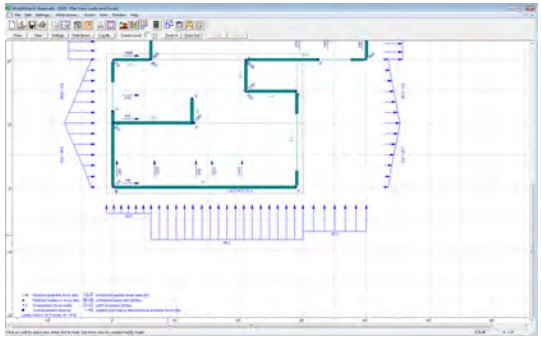


- A *wind C&C load* (wind suction on Components and Cladding) is listed beside the affected exterior wall as *Wind* C&C *Interior Magnitude/ End* Zone Magnitude.

- A *wind uplift load* is indicated by ⊙ at the two ends of a straight line drawn parallel to the affected exterior wall. The magnitude of the load is shown at mid-length of the line.
- *Dead loads and building masses* are indicated by ⊗ at the two ends of a straight line drawn parallel to the affected exterior wall. The magnitude of the load is shown at mid-length of the line. Dead loads are shown in blue while masses are shown in purple. Dead loads and building masses cannot be viewed at the same time.
- Uplift, dead and building mass point loads are indicated by single arrowheads.



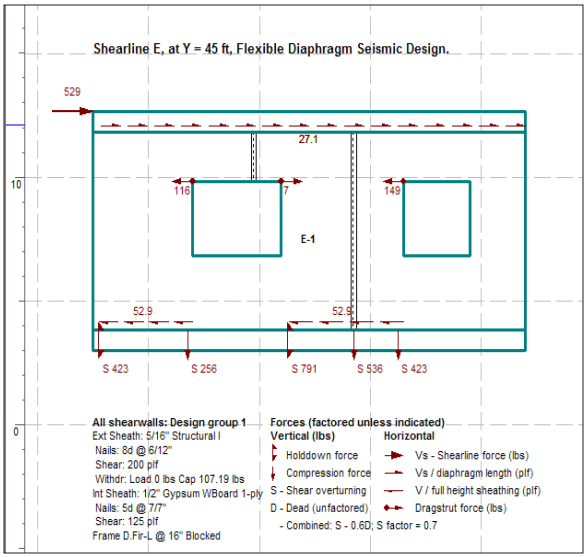
- Diaphragms are shown as purple rectangles that are numbered as F1, F2, etc. for each segment of the diaphragm for a given seismic load direction. The diaphragm segments are used for determining the horizontal distribution of seismic loads. Diaphragms can only be viewed in Generate Loads view and should only be viewed for one orientation at a time.
- *Hold-down forces* are indicated at the end of shearwall segments with a ◣ with the magnitude of the force adjacent to it and written diagonally to the shearwall direction. Plan view reports combined hold-down forces that indicate the combined affect of shear, uplift and dead loads (with the appropriate load reduction factors applied) at the hold-down location. If a compression load is present a single dot next to a "C" is shown on plans.



Elevation View

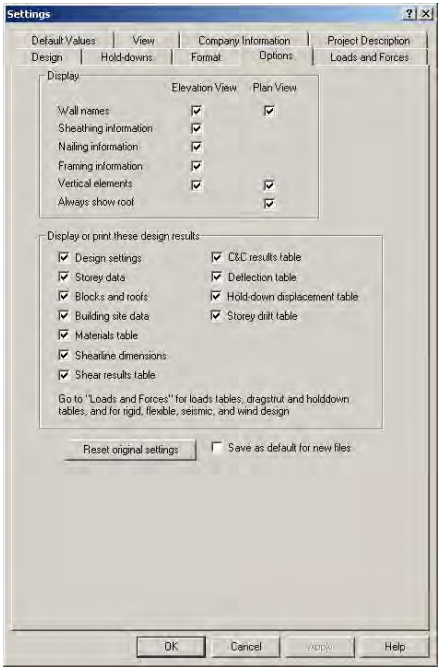
- A *wind and seismic shear* force is shown as horizontal single-headed shear flow arrows distributed along the top and bottom edges of affected shear walls. The total magnitude of the shear force for the shearline is shown, as are the magnitudes of the top and bottom shear flows accounting for openings. Also shown are the magnitudes of horizontal drag strut forces at the top of openings and vertical hold-down forces at the end of wall segments.
- A *wind C&C load* (wind suction on Components and Cladding) is listed beside the affected exterior wall as *Wind C&C Interior Magnitude/ End Zone Magnitude.*
- A *wind uplift load* is indicated by upward-pointing arrowheads distributed along the top edge of the affected wall. The magnitude of the load is shown at mid-length of the wall.
- *Dead loads and building masses* are indicated by downward-pointing arrowheads distributed along the top edge of the affected wall panels and openings. The magnitude of the load is shown at mid-length of the load. Dead loads and building masses cannot be viewed at the same time.
- Elevation View can report *hold-down* forces as Separate or Combined through the Show menu. Separate indicates the individual *shear* (S), *uplift* (U) and *dead load* (D) components, while *Combined* combines all of these affects together along with the appropriate load reduction factors applied.
  In the U.S. version, a compression force is indicated at the bottom corners of shearwall segments by downward-pointing arrows, with the magnitude adjacent to the arrows.

  A hold- down connection is depicted as two triangles, one on either side of the floor (to represent connector brackets).
- In Canada, an anchorage is depicted in the shape of an uppercase I (to represent a threaded rod with washers and nuts connecting the bottom plate of the upper wall to the top plate of the wall below).
- A *dragstrut force* is indicated by horizontal arrows acting on either side of and at the top of an opening. The magnitude of the force is shown directly under the arrow.



2.13 Results

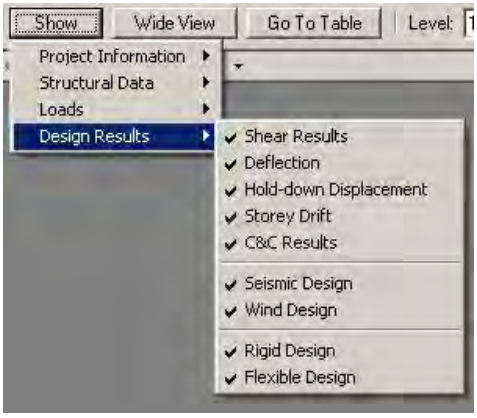
The *Results View* produces the design results report for a design run. This view is automatically opened when the user clicks on the *Run Design* action button and generates a new design results report. The report for the last design run is opened when the user directly clicks on the *Results View* action button. In both cases, the design must first be run to activate the Results View button. As well, the design must be re-run to produce an updated results report any time changes are made to the design data.

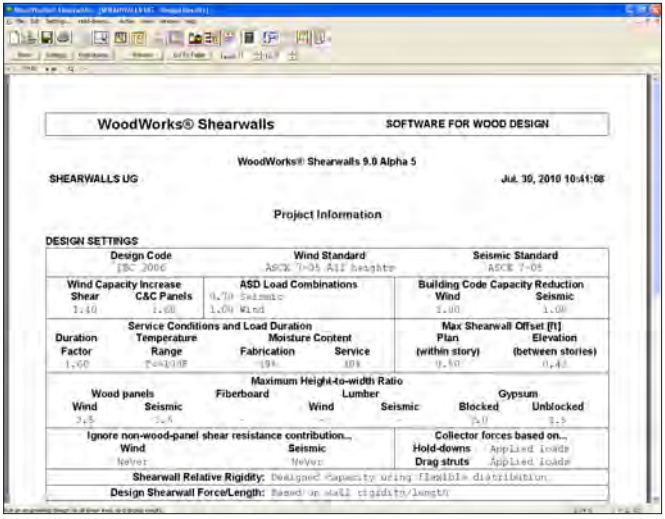


The design results report is divided into several sections, as described below. The sections which appear depend on the design option and the settings selected through the *Show button* or the *Settings* dialog (both the *Loads and Forces* and the *Option* menu items) boxes.

There is a lot of additional information in Help pertaining to generation and distribution of loads, and the related engineering assumptions used in the software.

The basic sections are *Project Information, Structural Data, Loads, Wind Design and Seismic Design.*





WoodWorks® Shearwalls SOFTWARE FOR WOOD DESIGN

WoodWorks® Shearwalls 9.0 Alpha 5

SHEARWALLS UG Jul. 30, 2010 10:41:08

Project Information

DESIGN SETTINGS

Design Code	Wind Standard	Seismic Standard
IBC 2006	ASCE 7-05 All heights	ASCE 7-05

Wind Capacity Increase		ASD Load Combinations	Building Code Capacity Reduction	
Shear	C&C Panels		Wind	Seismic
1.40	1.60	0.70 Seismic 1.00 Wind	1.00	1.00

Service Conditions and Load Duration				Max Shearwall Offset [ft]	
Duration Factor	Temperature Range	Moisture Content Fabrication	Service	Plan (within story)	Elevation (between stories)
1.60	T<=150F	19%	15%	0.50	0.40

Maximum Height-to-width Ratio					
Wood panels		Fiberboard	Lumber	Gypsum	
Wind	Seismic	Wind Seismic		Blocked	Unblocked
3.5	2.0	- -	-	2.0	3.5

Ignore non-wood-panel shear resistance contribution...		Collector forces based on...	
Wind	Seismic	Hold-downs	Applied loads
Never	Never	Drag struts	Applied loads

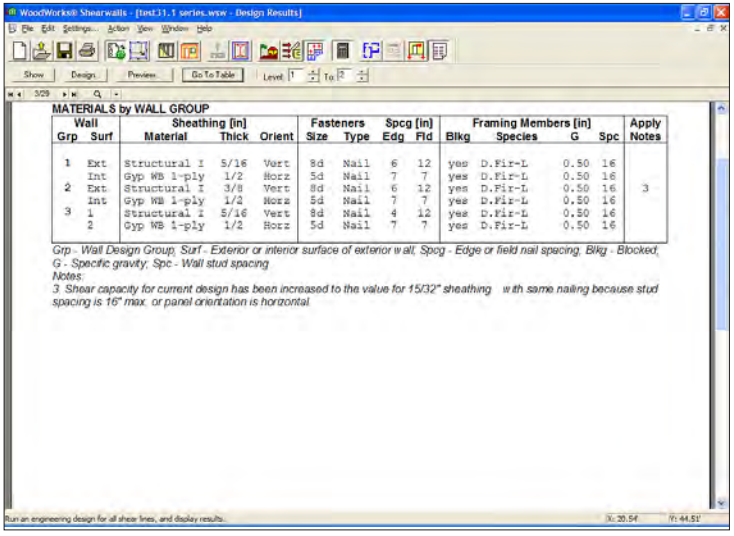
Shearwall Relative Rigidity: Designed Capacity using flexible distribution

Design Shearwall Force/Length: Based on wall rigidity/length

- The *Structural Data* section includes story data, design settings, site information, wall materials by wall groups, and shearline dimensions for the design.
- The *Loads* section includes the profiles, locations and magnitudes of wind shear, seismic, uplift and C&C loads, and direct forces, specified for the design.
- The *Wind Design* section contains the results of flexible and/or rigid diaphragm design, including shear results by shearline, dragstrut forces, hold-down design and deflection. It also provides component and cladding design results by shearline.
- The *Seismic Design* section contains the results of flexible and/or rigid diaphragm design, including shear results by shearline and dragstrut and hold-down forces.

Wall Groups

Shearwalls gathers wall assemblies of similar wall materials (sheathing, fastening and framing) into *Wall Groups* and assigns a wall group number to each group. This is summarized in the *Structural Data* section of the design results report.



WoodWorks® Shearwalls - [test11.1 series.wsw - Design Results]

MATERIALS by WALL GROUP

Wall Grp	Surf	Sheathing [in] Material	Thick	Orient	Fasteners Size	Type	Spcg [in] Edg	Fld	Framing Members [in] Blkg	Species	G	Spc	Apply Notes
1	Ext	Structural I	5/16	Vert	8d	Nail	6	12	yes	D.Fir-L	0.50	16	
	Int	Gyp WB 1-ply	1/2	Horz	5d	Nail	7	7	yes	D.Fir-L	0.50	16	
2	Ext	Structural I	3/8	Vert	8d	Nail	6	12	yes	D.Fir-L	0.50	16	3
	Int	Gyp WB 1-ply	1/2	Horz	5d	Nail	7	7	yes	D.Fir-L	0.50	16	
3	1	Structural I	5/16	Vert	8d	Nail	4	12	yes	D.Fir-L	0.50	16	
	2	Gyp WB 1-ply	1/2	Horz	5d	Nail	7	7	yes	D.Fir-L	0.50	16	

Grp - Wall Design Group, Surf - Exterior or interior surface of exterior wall, Spcg - Edge or field nail spacing, Blkg - Blocked, G - Specific gravity, Spc - Wall stud spacing

Notes:

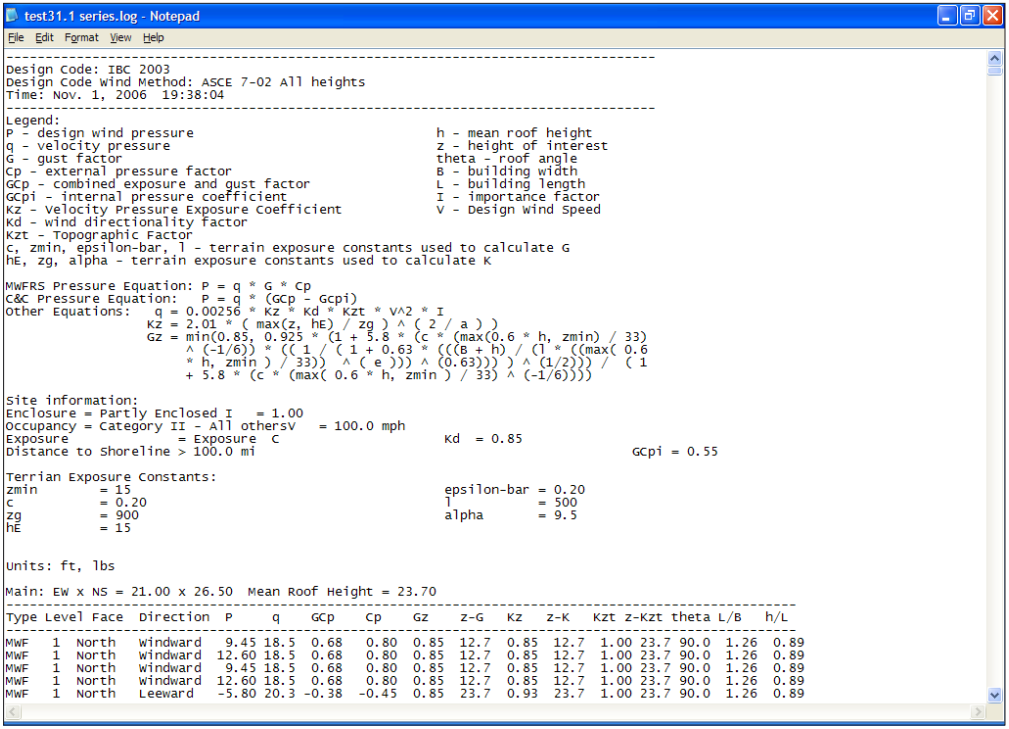
3 Shear capacity for current design has been increased to the value for 15/32" sheathing with same nailing because stud spacing is 16" max. or panel orientation is horizontal.

Wall group numbers are then referenced throughout the design results output report to avoid repeating the wall assembly materials information at several places in the output.

The design results can be saved as .pdf and .rtf files. For more information see section 1.3 "***Input and Output***".

Log File

Shearwalls creates a detailed log file of the intermediate calculations used to generate wind and seismic loads, and used in the rigid diaphragm analysis. These calculations are stored in a *.log ASCII text file that can be opened and viewed through any text editor, such as Notepad or Word. The log file also includes the definition and value of variables used in the generation of seismic and wind loads.



test31.1 series.log - Notepad

File Edit Format View Help

```
--------------------------------------------------------------------------------
Design Code: IBC 2003
Design Code Wind Method: ASCE 7-02 All heights
Time: Nov. 1, 2006  19:38:04
--------------------------------------------------------------------------------
Legend:
P - design wind pressure                          h - mean roof height
q - velocity pressure                             z - height of interest
G - gust factor                                   theta - roof angle
Cp - external pressure factor                     B - building width
GCp - combined exposure and gust factor           L - building length
GCpi - internal pressure coefficient              I - importance factor
Kz - Velocity Pressure Exposure Coefficient       V - Design Wind Speed
Kd - wind directionality factor
Kzt - Topographic Factor
c, zmin, epsilon-bar, l - terrain exposure constants used to calculate G
hE, zg, alpha - terrain exposure constants used to calculate K

MWFRS Pressure Equation: P = q * G * Cp
C&C Pressure Equation:   P = q * (GCp - Gcpi)
Other Equations:   q = 0.00256 * Kz * Kd * Kzt * V^2 * I
                Kz = 2.01 * ( max(z, hE) / zg ) ^ ( 2 / a ) )
                Gz = min(0.85, 0.925 * (1 + 5.8 * (c * (max(0.6 * h, zmin) / 33)
                     ^ (-1/6)) * (( 1 / ( 1 + 0.63 * (((B + h) / (l * ((max( 0.6
                     * h, zmin ) / 33))  ^ ( e ))) ^ (0.63))) ) ^ (1/2))) /  ( 1
                     + 5.8 * (c * (max( 0.6 * h, zmin ) / 33) ^ (-1/6))))

Site information:
Enclosure = Partly Enclosed I   = 1.00
Occupancy = Category II - All othersV   = 100.0 mph
Exposure          = Exposure  C                   Kd  = 0.85
Distance to Shoreline > 100.0 mi                                  GCpi = 0.55

Terrian Exposure Constants:
zmin        = 15                                  epsilon-bar = 0.20
c           = 0.20                                l           = 500
zg          = 900                                 alpha       = 9.5
hE          = 15


Units: ft, lbs

Main: EW x NS = 21.00 x 26.50  Mean Roof Height = 23.70
--------------------------------------------------------------------------------
Type Level Face  Direction  P     q    GCp    Cp    Gz    z-G   Kz   z-K   Kzt z-Kzt theta L/B   h/L
--------------------------------------------------------------------------------
MWF   1  North   Windward   9.45 18.5  0.68   0.80  0.85  12.7  0.85  12.7  1.00 23.7 90.0  1.26  0.89
MWF   1  North   Windward  12.60 18.5  0.68   0.80  0.85  12.7  0.85  12.7  1.00 23.7 90.0  1.26  0.89
MWF   1  North   Windward   9.45 18.5  0.68   0.80  0.85  12.7  0.85  12.7  1.00 23.7 90.0  1.26  0.89
MWF   1  North   Windward  12.60 18.5  0.68   0.80  0.85  12.7  0.85  12.7  1.00 23.7 90.0  1.26  0.89
MWF   1  North   Leeward   -5.80 20.3 -0.38  -0.45  0.85  23.7  0.93  23.7  1.00 23.7 90.0  1.26  0.89
```

2.14 Perforated Shearwalls (U.S. Only)

FIGURE 1. Typical Shearwall Composed of Three Traditional Shearwall Segments

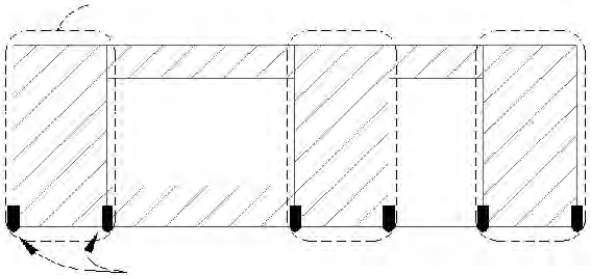
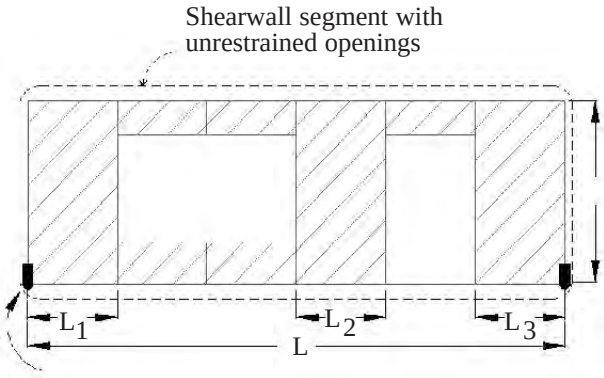


FIGURE 2. Typical Perforated Shearwall with Unrestrained Openings



Traditional Shearwalls (Type I)

Wood frame shearwalls are traditionally designed using shearwall segments that extend the full-height of the wall.

Shearwalls containing openings are treated as multiple shearwall segments, as shown in Figure 1. Each segment requires fasteners to transfer shear and provide overturning restraint resulting in additional holddowns per wall. The design capacity of the shearwall is assumed to equal the sum of the capacities of each shearwall segment. Sheathing above and below openings typically is not considered to contribute to the overall performance of the wall.

Perforated Shearwalls (Type II)

Another approach considers a shearwall segment with openings, which is illustrated in Figure 2. Here, the wall is identical to the shearwall in Figure 1, but without intermediate overturning restraints next to the openings. This condition is considered a shearwall segment with unrestrained openings (perforated or Type II shearwall). The capacity is clearly less than would be expected from the same shearwall containing multiple shearwall segments, each fully restrained against overturning. However, the capacity of the wall shown in Figure 2 may be sufficient for a particular application while reducing the total number of overturning connections.

Method

Finding the design shear capacity involves relating the strength of a shearwall with unrestrained openings to a similar shearwall consisting of traditional shearwall segments. Empirically derived adjustment factors that relate the strength of shearwalls with unrestrained openings to similar shearwalls containing multiple traditional shearwall segments are presented in Table 1. The perforated shearwall procedure calls the adjustment factor the "opening adjustment factor." This is refered to as the *Perforated Factor* in the Result output.

Opening adjustment factors (refered to as Perforated Factor in the Results output) for a range of typical opening heights are provided in Table 1 for eight and ten-foot high walls. Factors for other wall heights can be determined from the ratio of opening height to wall height.

Table 1 — Opening Adjustment Factor for Perforated Shearwalls

	Maximum Unrestrained Opening Height (Window or Door Height)				
	H/3	H/2	2H/3	5H/6	H
8' Wall		2'-8"	4'-0"	5'-4"	6'-8"
10' Wall		3'-4"	5'-0"	6'-8"	8'-4"
Percent Full-Height Sheathing	Effective Shear Capacity Ratio				
0%	1.00	0.67	0.50	0.40	0.33
10%	1.00	0.69	0.53	0.43	0.36
20%	1.00	0.71	0.56	0.45	0.38
30%	1.00	0.74	0.59	0.49	0.42
40%	1.00	0.77	0.63	0.53	0.45
50%	1.00	0.80	0.67	0.57	0.50
60%	1.00	0.83	0.71	0.63	0.56
70%	1.00	0.87	0.77	0.69	0.63
80%	1.00	0.91	0.83	0.77	0.71
90%	1.00	0.95	0.91	0.87	0.83
100%	1.00	1.00	1.00	1.00	1.00

Full-Height Sheathing

The percentage of full-height sheathing (FHS) is based on the ratio of full-height sheathing length to shearwall segment length. To be considered full-height sheathing, the length of each full-height sheathed section must equal or exceed the prescribed height-to-length ratios for shearwall segments in the applicable building code.

Values for 0% full-height sheathing are provided for interpolation purposes only. The 100% full-height sheathed case represents a traditional shearwall segment with a corresponding opening adjustment factor of one.

Shear connections

Shear connections capable of resisting the allowable unit shear of a traditional shearwall segment are required throughout the segment. While the actual shear connection requirements may be less when using the perforated shearwall method, traditional shearwall segment shear connector requirements account for the non-uniform distribution of shear loads in the perforated wall.

References

Diekmann, E. 1992. Designing for wind or seismic loads. Proceedings of the International Seminar on Wood Engineering.

Dolan, J.D.; Johnson, A.C. 1996. Monotonic and cyclic tests of shearwalls with openings. Blacksburg, VA: Virginia Polytechnic Institute and State University. Timber Engineering.

Douglas, Bradford; Sugiyama, Hideo. 1995. Perforated shearwall design approach. Washington, DC: American Forest & Paper Association.

Standard Building Code (SBC) Wood Section 1994 edition with 1996 revisions. Birmingham, AL: Standard Building Code Congress International.

Sugiyama, Hideo. 1981. The evaluation of shear strength of plywood-sheathed walls with openings. Mokuzai Kogyo (Wood Industry). 36-7, 1981.

WFCM-96 Wood Frame Construction Manual for One- and Two- Family Dwellings — 1995 SBC High Wind Edition. Washington, DC: American Forest & Paper Association. 235 p.

2.15 Shearwall Segments Without Hold-downs (Canada Only) (excerpt from Wood Design Manual 2001)

Traditionally, shearwalls have been designed using chords and hold-down connections at the ends of all shearwall segments. Hold-downs are designed to transfer the chord segment overturning force, T_f, to the shearwall or foundation below. CSA O86 includes provisions for design of shearwall segments without hold-down connections.

Without hold-downs, the overturning tension force is transferred from the top wall plate to the bottom wall plate through the shearwall sheathing. Since a portion of the shearwall sheathing is used to resist the overturning force, the shear capacity of the sheathing is reduced. Even though hold-down anchors are not used, anchorage is still required to transfer the uplift force from the wall plate to the foundation or shearwall below (see Commentary to CSA O86-01)

In some cases, hold-down connections may be placed at one end of a shearwall segment. Where the load is from the end of the segment that contains the hold-down, overturning uplift forces will be resisted by the hold-down. When the load is from the direction opposite the hold-down, the sheathing resists over-turning uplift forces. The shear capacity of the shearwall segment is multiplied by the hold-down factor, J_{hd}, which must be calculated for loads acting in opposite directions.

Hold-down connections are not required for shearwall segments where the specified strength is adjusted by the J_{hd}, factor, and the following conditions are met:

- the sum of the factored basic shear resistances on both sides of the shearwall, V_{hd}/L_W, < 10.3 kN/m;
- shear resistance of the wood-based panel, v_r, is based on nail diameters ≤ 3.25 mm; and edge panel nail spacing -≥ 100mm;
- the maximum height of the shearwall, H_s, is 3.6m; and
- the factored uplift restraint force at the bottom of the end stud of the shearwall segment, P_j, is ≥ 0.

J_{hd} is calculated as:

Case 1

J_{hd} = 1.0 where there is sufficient dead load to resist overturning uplift forces or hold-down connections resist all of the overturning force.

Case 2

Where there is no net uplift at the top of the shearwall segment due to overturning and there is no hold-down connection at the bottom of the shearwall segment to resist overturning:

$$J_{hd} = \sqrt{1 + 2\frac{P_j}{V_{hd}} + \left(\frac{H_s}{L_W}\right)^2 \frac{H_s}{L_W}} \leq 1.0$$

Where:

P_j = factored uplift restraint force at the bottom of the end stud of the segment calculated as shown below (kN)

V_{hd} = sum of the factored basic shear resistances on both sides of the shearwall segment calculated with J_{hd} = 1.0 (kN)

$= \Sigma(v_r J_n J_{ub} + v_{rg}) \times L_W$

H_s = height of the shearwall segment measured from the bottom of the bottom plate to the top of the top plate (m)

L_w = Length of the shearwall segment (m)

Case 3

Where hold-downs are provided at the bottom wall plate to resist overturning forces but the sheathing is in tension at the top of the shearwall segment due to overturning forces from upper storeys.

$$J_{hd} = \frac{V_{hd} + P_t}{V_{hd}} \leq 1.0$$

where:

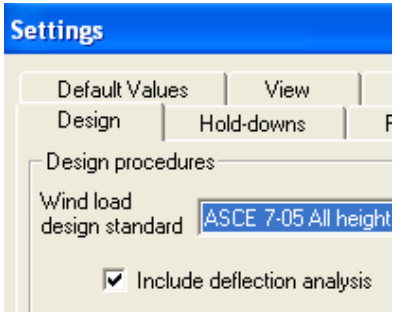
P_t = factored uplift restraint force at the top of the end stud of the segment calculated as shown below (kN)

Note: $P_t < 0$

V_{hd} = sum of the factored basic shear resistances on both sides of the shearwall segment calculated with J_{hd} = 1.0 (kN) (see above)

2.16 Deflection of Shearwalls

Shearwalls calculates the deflection of each wall segment between openings for each design case. It uses this deflection to determine the storey drift and check that the drift is within allowable limits for seismic design. Shearwalls also distributes loads to shearlines and within shearlines to segments based on equalized deflection of segment. That is, rather than using the capacity of the shear walls to approximate the rigidities for distributing loads to and within shear lines, the software includes the option to determine rigidities based on deflection-derived stiffness. The option to ***Use shearwall deflection to calculate rigidity*** is available only if the ***Settings / Design***: ***Include deflection analysis*** is selected.

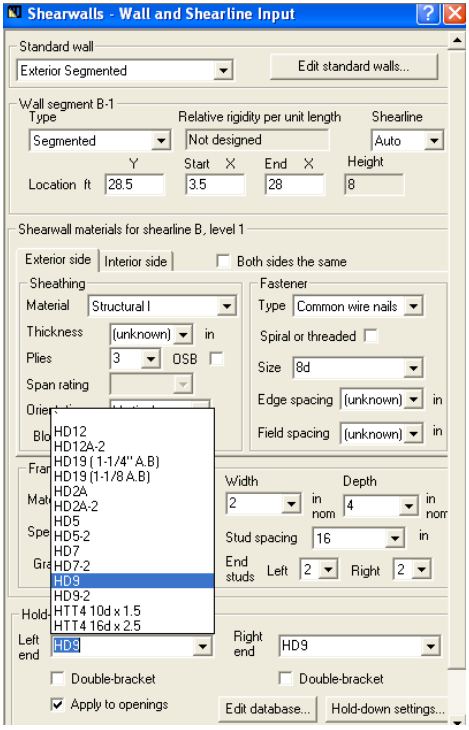


While deflection analysis is considered more accurate in certain circumstances, the added iterative calculations running in the background can slow the run time of Shearwalls. For example, it would not be unusual for a design run to take 3 times longer, in the order of several minutes, when deflection analysis is selected and the distribution of loads is based on rigidity derived from deflection.

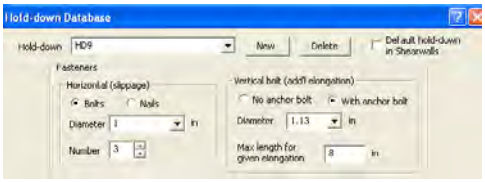
Hold-downs

The ability to input hold-down connectors to a hold-down database for use in design for overturning forces and for deflection analysis gives added flexibility to Shearwalls. Previously Shearwalls reported hold-down forces at each hold-down location, but did not specify the hold-down connections used.

Hold-downs button
This button opens the hold-down database window where hold-downs can be created and used in the design. Several hold-downs are already created and can be chosen from a drop0down list. The hold-downs specified in the database are selected in the *Wall and Shearline* Input as well as the *Opening* Input at each hold-down location for use in determining the component of deflection attributable to the wall anchorage system's slip.



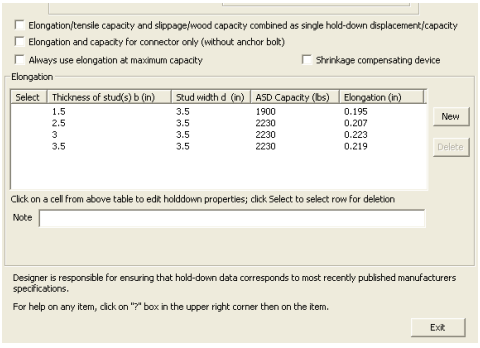
Horizontal and vertical fastener information as appropriate for the hold-down is entered in the fastener section.



There are four check boxes that allow the user to specify what will be taken into account for the displacement.

- *Elongation and slippage combined as single hold-down displacement*: If this is checked, the displacement value entered in the table below includes displacement from all sources of deflection related to the connection assembly, including the elongation of the connector itself, the horizontal fastener slippage used to connect the hold-down bracket to the studs, and the vertical bolt elongation up to the length of vertical bolt shown in the top right of this input form. Manufacturers typically include all these sources of deflection in their tables, therefore selecting this checkbox will be typical. If it is not checked, the value indicates the elongation only and therefore the slippage will be calculated separately.

- *Elongation for connector only (without anchor bolt):* When this box is checked, the elongation value does not represent the anchor bolt elongation, which is calculated using the bolt length indicated in the *Structure* dialog. This option is not viable when the elongation and slippage are combined as single hold-down displacement.

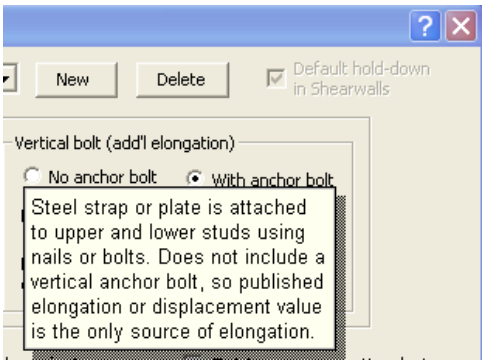


- *Shrinkage compensating device:* If the hold-down is capable to adjust to the wood movement, this box should be checked so the wood shrinkage and crush component are not included in the displacement.

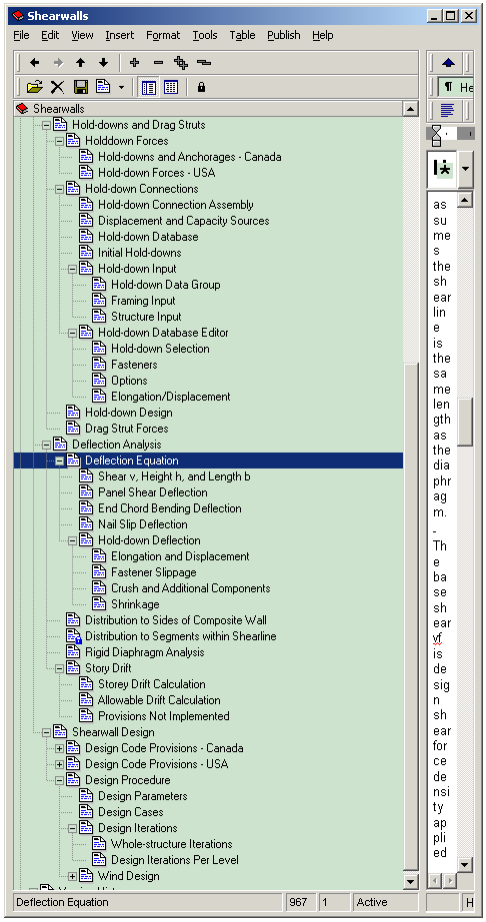
- *Always use elongation/displacement at maximum capacity:* If this box is checked, the displacement is assumed to correspond to the allowable hold-down capacity even if the actual force at the hold-down is less than the listed capacity of the hold down. For example, if *Always use elongation/displacement at maximum capacity* is checked, and if the calculated uplift force that a hold-down is required to resist is 1000 lbs, but the selected hold-down has a capacity of 2000 lbs with a corresponding displacement of 0.20 inches, the hold-down elongation or displacement will be based on the listed displacement value of 0.20 inches associated with the maximum 2000 lb capacity. Left unchecked, the displacement will be based on the ratio of actual force : capacity, in this case 50% of 0.20 inches, or 0.10 inches.

The deflection of shearwall is important to ensure it remains within the allowable storey drift limits. It also determines the rigidity of the shearwall segments for load distribution within shearlines, and the rigidity of the shearline for use in the rigid diaphragm distribution method.

For more information, use the context sensitive help found on this *Hold-down database input* form by selecting the **?** symbol at the top right and clicking on the area for which help is needed.

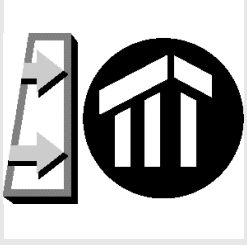


For more detailed information regarding deflection and hold down related information refer to **Help** and search for keywords as shown below. Also, refer to the **Shearwalls - New Features** file located in the main WoodWorks folder.



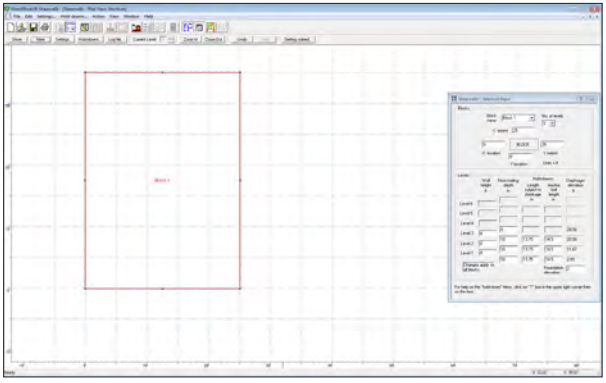
Shearwalls Tutorial

B.3

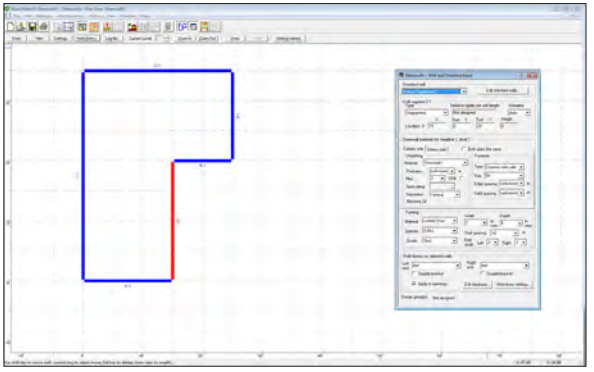


3.1 Quick Start Tutorial

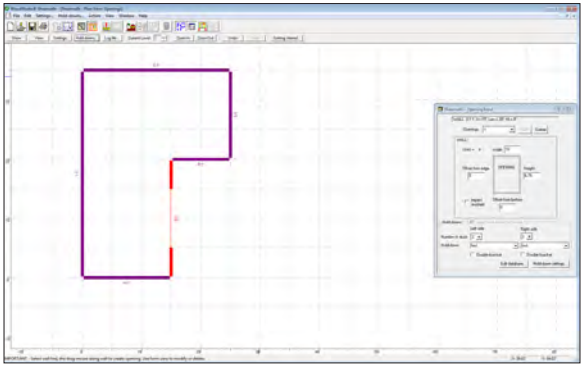
This tutorial will quickly provide you with the major steps and procedures required to design a wood-frame building for manually applied lateral loads. See the Advanced Tutorial for a more complete design example with automatically generated loads.



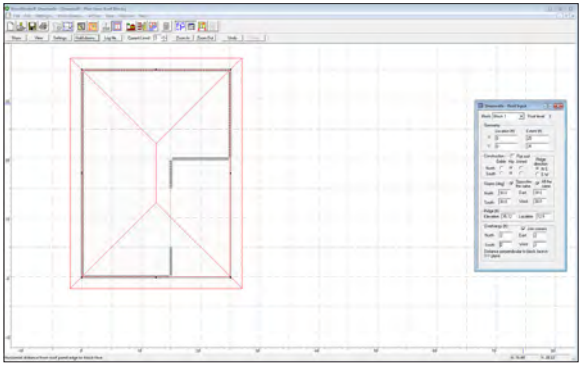
1. Start Shearwalls. The Structure button is highlighted and the ***Structure Input*** form is active.
2. Define a structure block by clicking & holding the left mouse button while the cursor points at approximately ***X=0 & Y=0*** and dragging the cursor to approximately ***X=25 & Y=35***. You may need to manually adjust the ***Location and Extents*** in the ***Structure Input*** form to define the structure block.
3. Increment ***No. of Levels*** in the ***Structure Input*** form to **3 levels** for the selected structure block.
4. Click the ***Walls*** button and highlight wall **2-1**. You may change the length, location, type or material of the wall in the ***Wall Input*** form.
5. Split wall 2-1 by pointing the cursor at (25,0) holding down the left mouse button while dragging the cursor to (25,20), then releasing the left mouse button. Walls 2-1 and 2-2 will now be created.



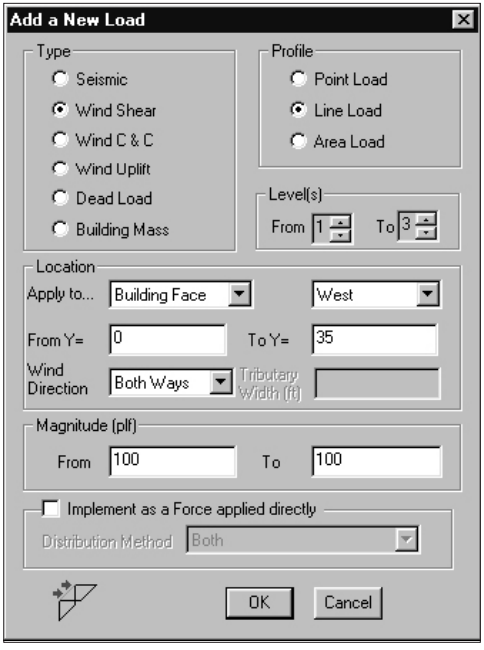
6. Move wall 2-1 by pointing the cursor at (25,10) holding down the shift key & left mouse button while dragging the cursor to (15,10), then releasing the left mouse button & Shift key.



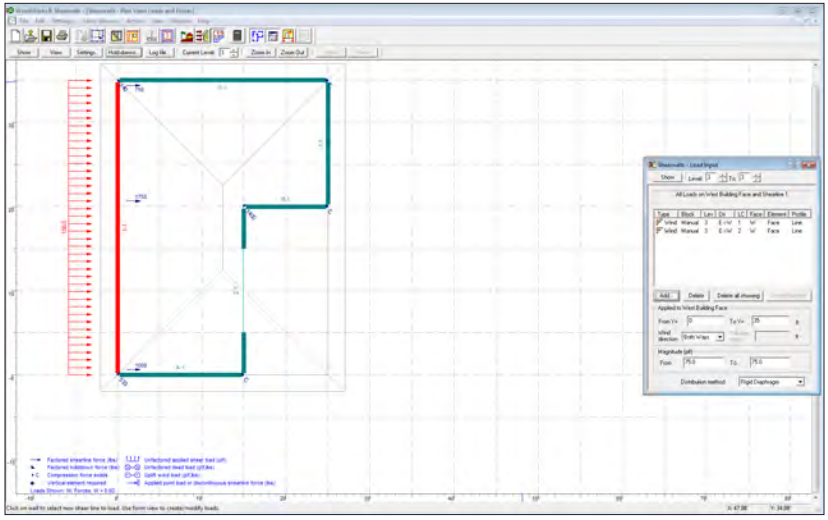
7. Click the ***Openings*** button, point the cursor at (15,5), hold down the left mouse button & drag the cursor to (15,15) and release the left mouse button. You may change the offsets, width and height of the opening in the ***Opening Input*** form.

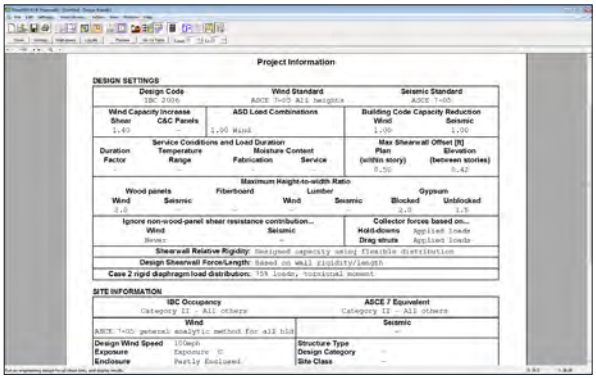


8. Click on the ***Extend*** Levels button. This will automatically create the levels 2 and 3 as a copy of level 1.
9. Click on the ***Roof Blocks*** button. In the ***Roof Input*** form, under **Construction** select **Hip** for the north and south sides of the building, and enter overhangs of 2ft for all sides of the building.

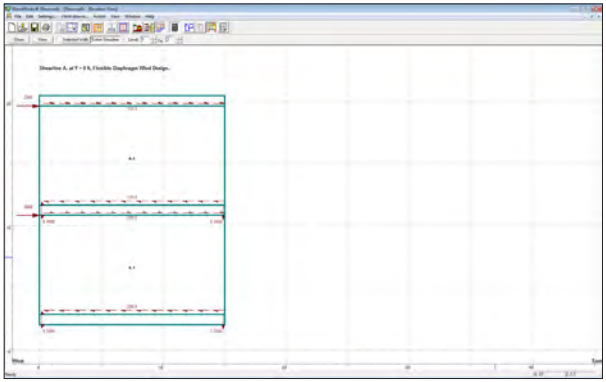


10. Click on the ***Generate Loads...*** button to automatically generate wind or seismic loads. Loads will be added manually for this example, therefore do not generate loads and proceed to step 11.
11. Click on the ***Loads & Forces*** button, click ***Add…***, select **Wind Shear, West Building Face, Line Load**, apply from level 1 to 3, Magnitude = 100 plf and click **OK**.





12. Click on the ***Run Design*** button and view the ***Results*** report. You can navigate directly to the desired table using the ***Go To Table*** button.



13. Click on the ***Loads and Forces*** button, highlight wall A-1 and click on the ***Elevation View*** button to see the load distribution graphically on a shearwall or entire shearline.

14. If you close an Input form, click on the ***Forms*** button to restore the appropriate form.

3.2 Advanced Tutorial

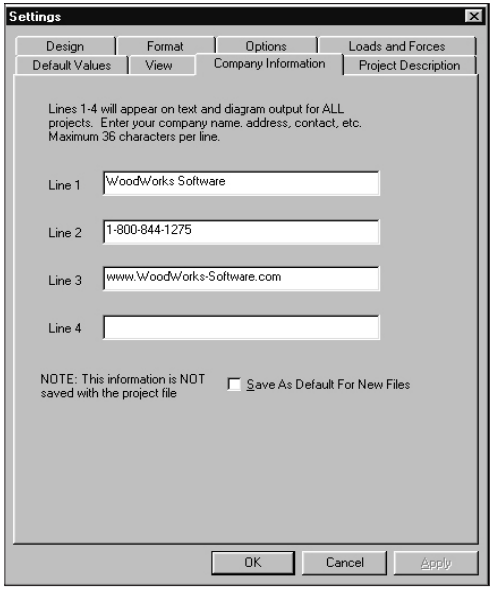
In this tutorial you will create a two-story wood-frame structure with a one-story garage and design it for automatically generated wind and seismic loads.

To begin, start the Shearwalls program or select ***File/New*** to start a new file if you are already in Shearwalls.

Settings
Define the initial settings and preferences for your project.

Company Information

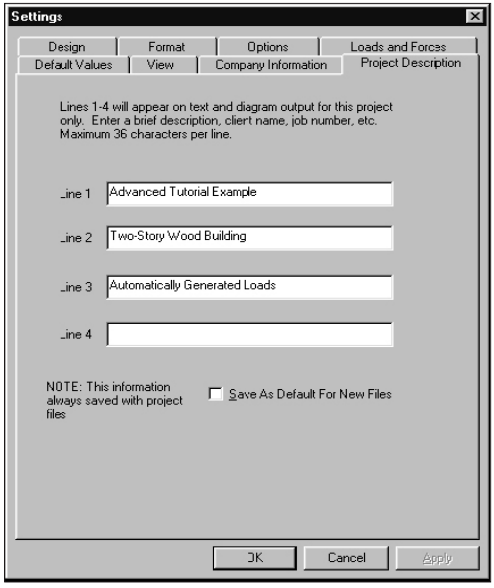
1. From the ***Settings*** menu, select the ***Company Information*** tab.
2. Enter your company information.



Check out our online video called "***Advanced tutorial from User Guide***" to see this advanced tutorial demonstrated.

Project Description

1. Now click on the **Project Description** tab.
2. Enter the relevant project description.



View Options

1. Click on the ***View*** tab.
2. Set the snap increment to ***6 inches***.
3. Set ***Fit View Area*** to ***Window*** and ***Fit Building*** to ***View Area*** to off.
4. Set the ***Display Gridlines…*** to off

Design Options

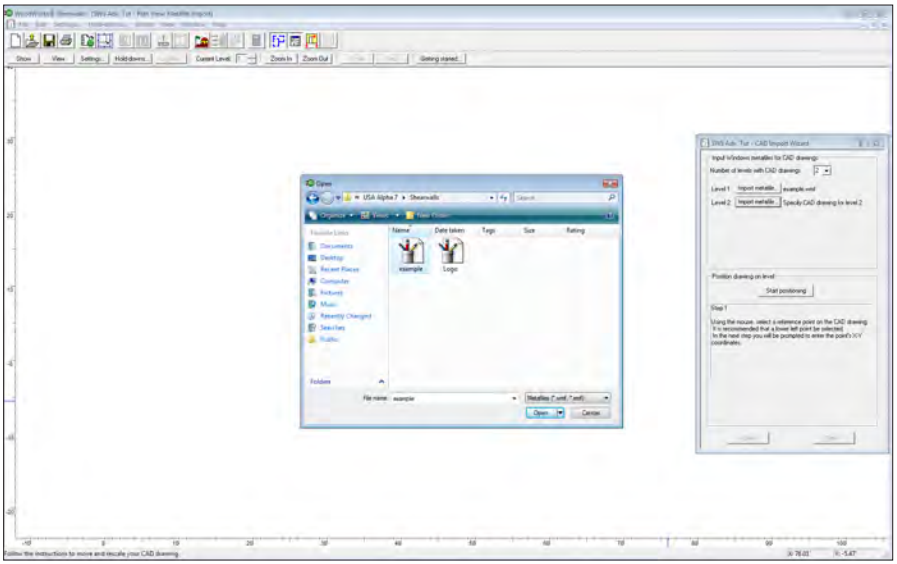
1. Click on the ***Design*** tab.
2. Click on ***Reset Original Settings***.
3. Select ***appropriate Wind load design standard.***

To save any of the Settings information as the default, click on the Save as Default for New Files option for the appropriate tab.

4. Click ***OK***.

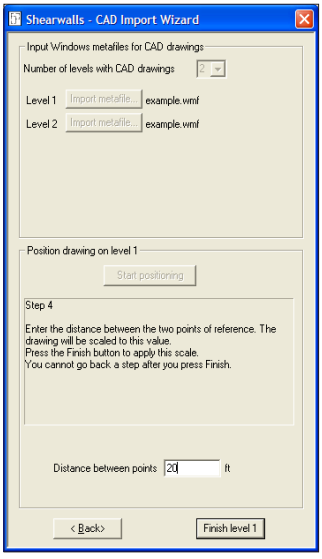
CAD Import

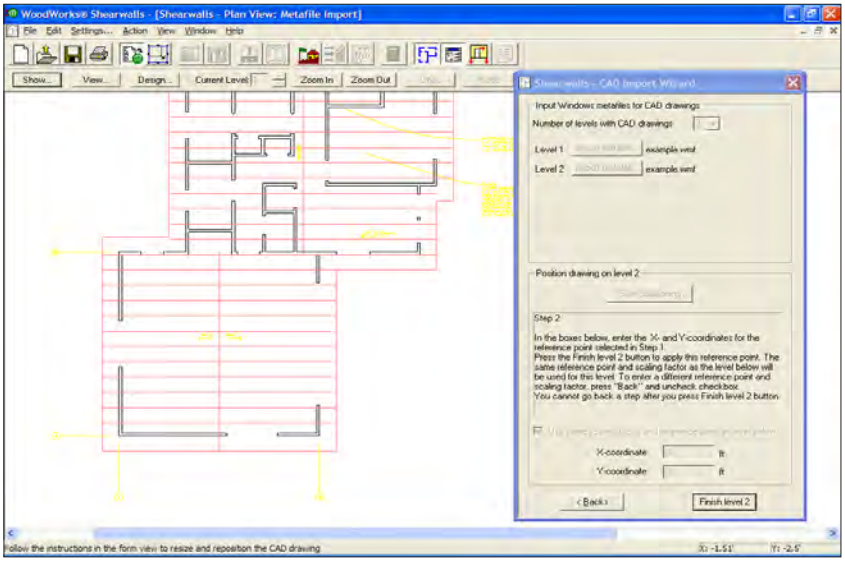
1. Click on the ***Import CAD*** button and select Number of Levels =2.
2. Click ***Import Metafile*** and from the *Examples* subdirectory under the Shearwalls directory, select the ***example.wmf*** file and click ***Open***. Repeat for second level.
3. Following the ***Import Wizard*** instructions, select the ***Start Positioning*** button and enter the first reference point as the bottom left hand corner of the building using the mouse.

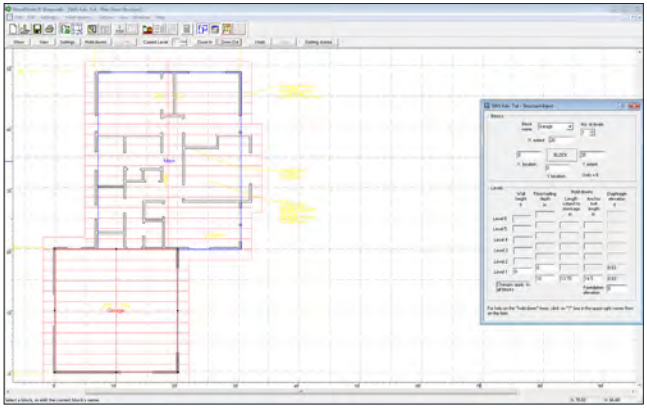


Note: For multi-level buildings with varying layouts on each level, importing different metafiles for each level can be done.

4. Click ***OK***.
5. Select the bottom right corner of the building as the second reference point.
6. Enter 20(ft) as the distance between the two reference points.
7. Click ***Finish level 1***.
8. Continue with *CAD Import Wizard* scaling instructions for level 2.







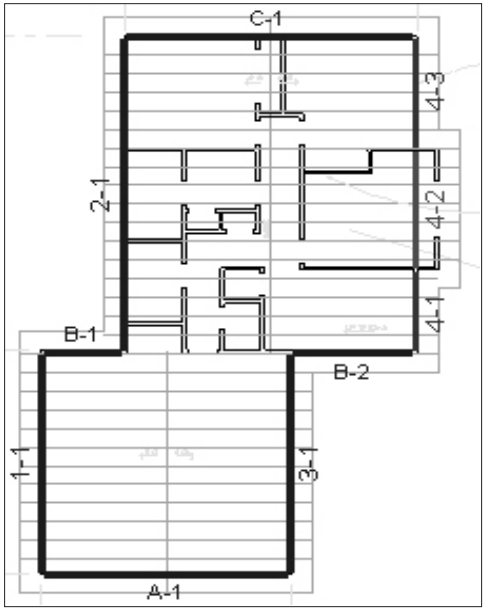
Structure Blocks & Levels

1. Click on the ***Structure*** button.
2. Use the ***Zoom In*** and ***Zoom Out*** buttons to adjust the CAD building image to fill the viewing area.
3. Layout the first ***Structure Block*** along the exterior walls by clicking and dragging a rectangle from the top right hand corner of the structure to the only inside corner on the left hand side of the building. With the block still highlighted, enter ***Main*** in the Block Name field and select ***2*** as the number of levels. This block will represent the two-story living space of the structure.
4. Layout a second structure block representing the garage portion of the structure by clicking and dragging a rectangle from the bottom left hand corner of the structure to the inside corner on the right hand side of the building where the garage ends. Blocks must abut or over lap slightly. Enter ***Garage*** as the block name and select ***1*** as the number of levels.
5. Set the foundation elevation as ***3*** ft.

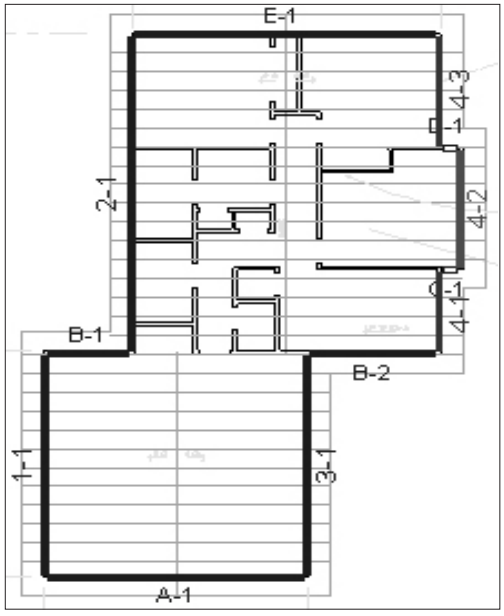
1st Story

Walls

1. Click on the ***Walls*** button.
2. Extend outward the main entrance of the structure. Click on wall 4-1 and click and the mouse from a point on wall 4-1 where the main entrance extension first begins to where it ends. There should now be three walls along shearline 4, including 4-1, 4-2 and 4-3.

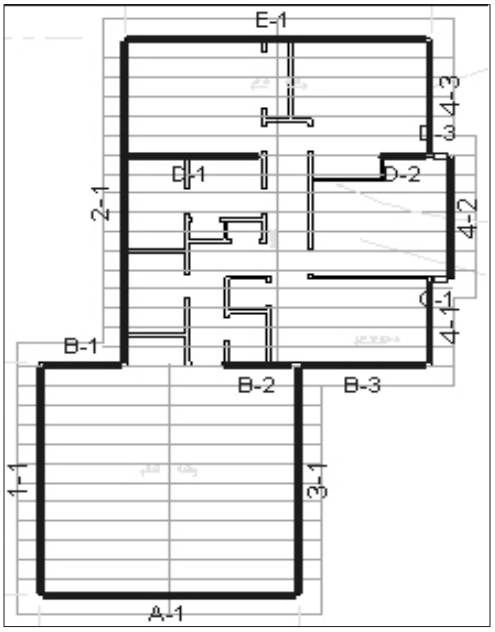


3. Move wall 4-2 to the right to extend the main entrance and match the CAD file by holding the Shift key and clicking on wall 4-2. While holding the left mouse button down, move the arrows to the new position and release.

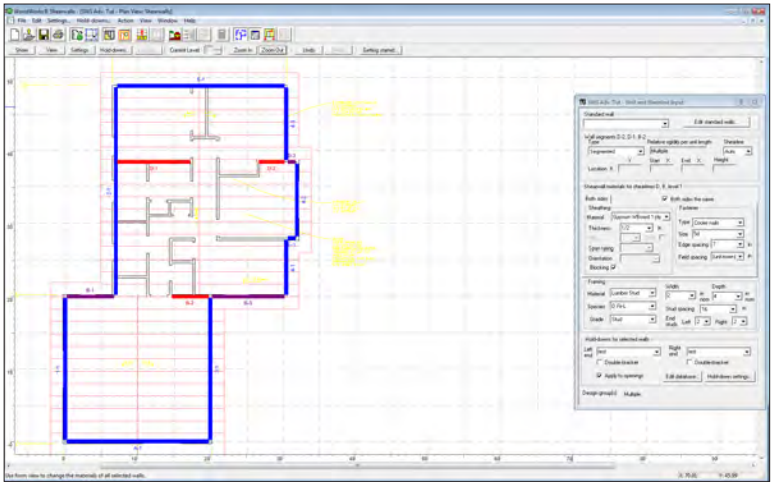


4. Continue this process until the building footprint matches the CAD file as shown.

5. Add three interior walls by clicking at the start of each wall and dragging the mouse to the end of the wall.

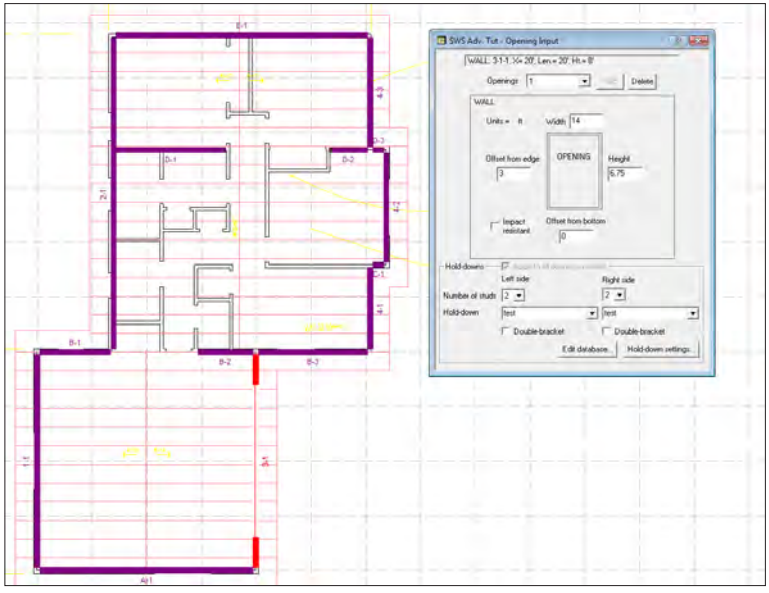


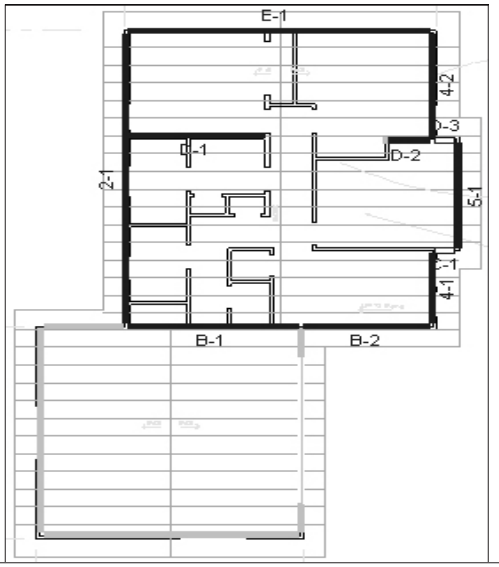
6. Select all three interior walls by holding CTRL and clicking on walls.
7. Change from *Exterior Segmented* to GWB 1 ply both sides same in *Wall and Shearline Input.*



Openings

1. Click on the ***Openings*** button.
2. Create a garage door opening by selecting wall 3-1 and clicking on a point offset 3 ft from the edge of the building and dragging a 14(ft) opening. This can also be achieved by directly entering input in the ***Opening Input*** form.
3. Continue to place door and window openings around the structure as desired.
4. To ease viewing, click on the ***Import CAD*** button to remove the CAD image from the background. This can be turned back on at any time.



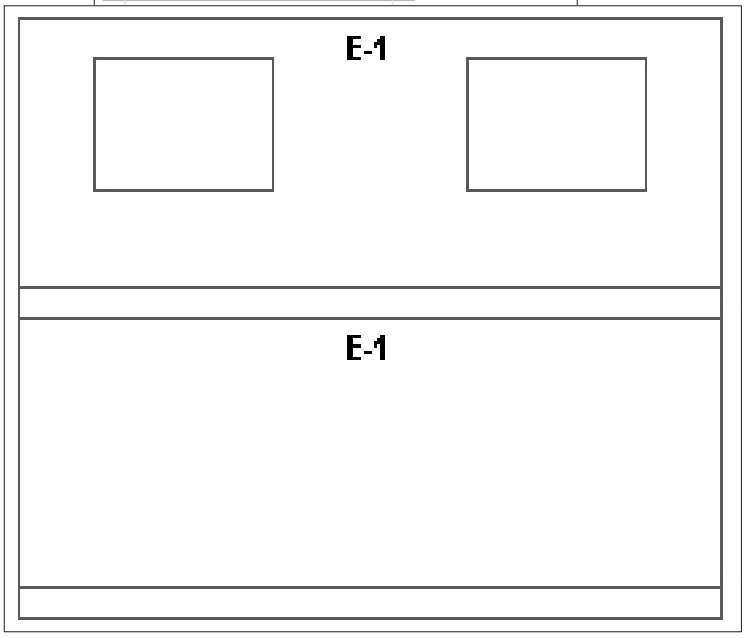


2nd Story

1. Click on the ***Extend Walls*** button. This automatically creates the upper levels for all multi-leveled blocks. In this case the second level is created for the Main structure block.
2. The current level should be level 2.

Walls

1. Click on the ***Walls*** button. Add two interior walls on the level 2 directly above the interior walls on level 1, as shown.

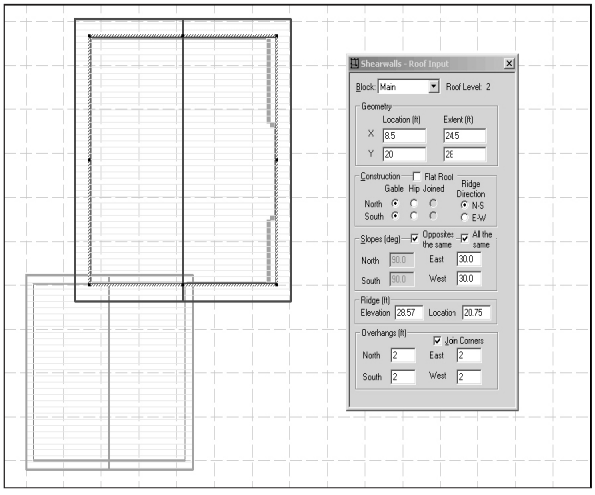


Openings

1. Click on the ***Openings*** button.
2. Select wall E-1 and create two small 6(ft) wide window openings with a 3.75(ft) height and 3(ft) offset from the bottom.
3. To view the current openings, select wall E-1 and click on the Elevation button.
4. Select level 1 to 2 in the data bar.

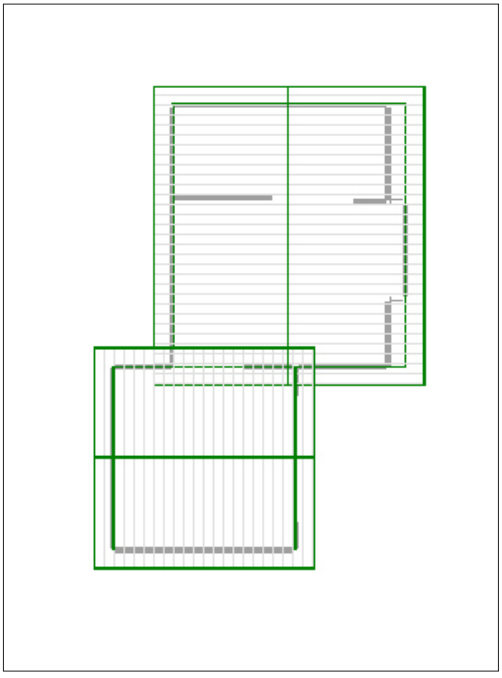
Roof Blocks

1. Click on the ***Roof Blocks*** button. Roof blocks will automatically be assigned to all structure blocks having an initial geometry based on the defaults set in **Settings/ Default Values.**
2. Select the Main block. Set the east and west slopes of the roof at 30 degrees and set overhangs of 2 ft. on all sides of the roof.



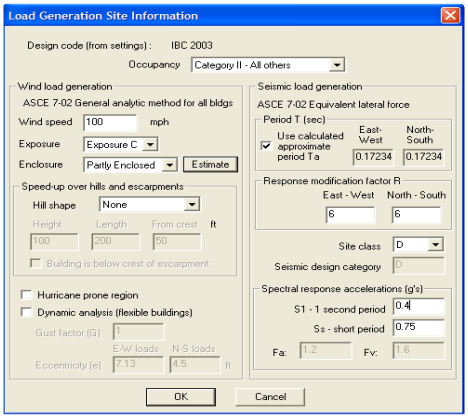
3. Select the Garage block. Set the ridge direction to E-W, set the north and south slopes of the roof at 30 degrees and set overhangs of 2 ft. on all sides of the roof.

4. Click on the ***Import CAD*** button to turn off the CAD image in the background.



Generating Loads

1. Before generating loads, click on the ***Site Information*** toolbar icon. From the Load Generation Site Information input dialog you can set the parameters to be used to generate wind and seismic loads. To change the Design Code selection, go to the Settings/Design tab.



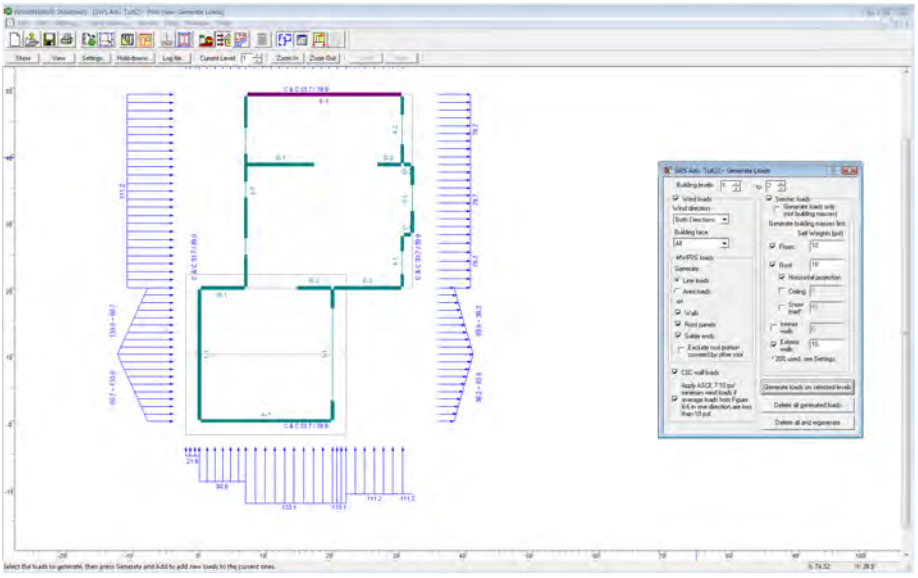
2. Close Site Information.

Note: You can limit load generation to specific levels by selecting only the range of floors. Skipping floor load generation can be achieved by selecting the ***"Loads and Forces"*** screen, select the floor for which you do not want loads generated and ***"Delete all Showing.***"

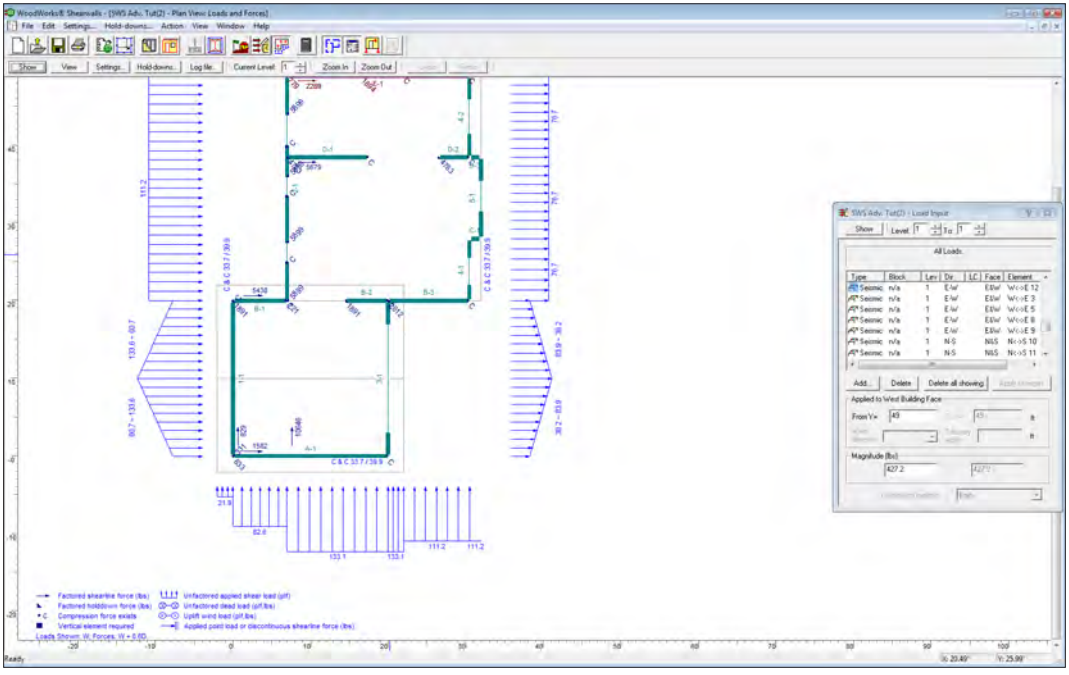
3. Click on the ***Generate Loads*** toolbar icon.

4. Select Wind and Seismic loads to be generated from level 1 to 2. All other setting should remain as shown.

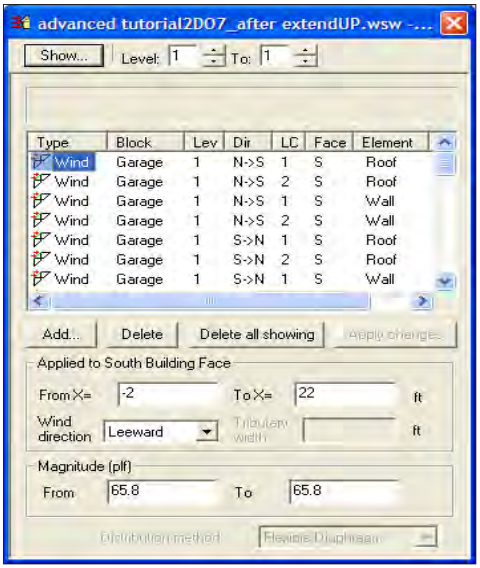
5. Click on ***Generate Loads on Selected Levels*** to automatically generate loads.



6. Use the ***Show*** button menu in the data bar to turn on and off viewing of a variety of data.
7. To view seismic loads, click on ***Show/Seismic/Seismic Loads and Forces***. Go to ***Show/Orientation*** to view an alternative load orientation.

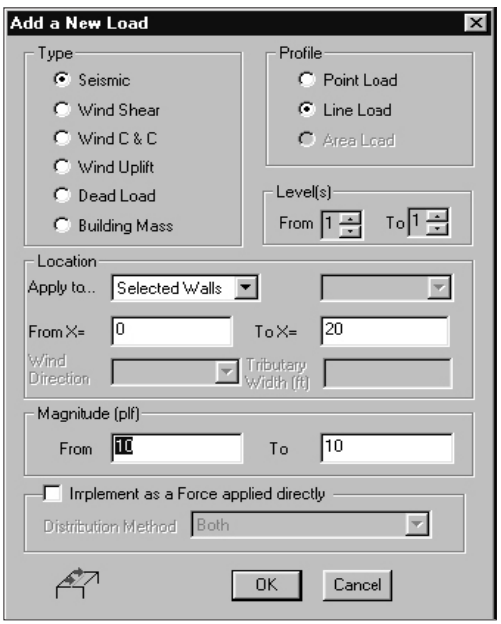


Note: Loads shown based on seismic, R=6, flexible diaphragm, and may not be identical to results using the latest ASCE 7 / IBC.

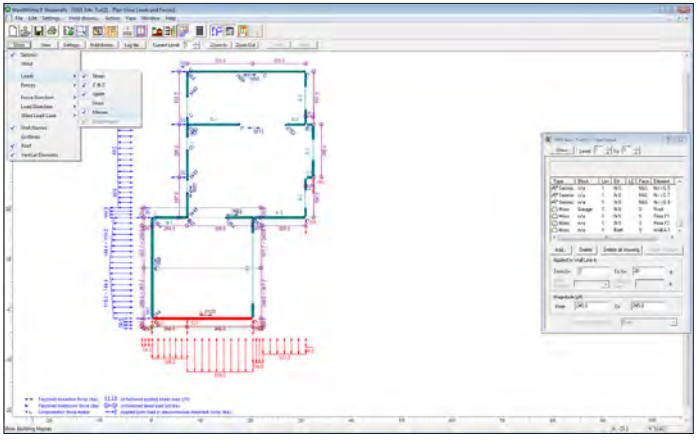


Modifying & Manually Applying Loads

1. Click on the ***Loads and Forces*** button.
2. Select the south wall (A-1) on level 1. All generated loads acting on wall A-1 will appear in the ***Load Input*** form.
3. Generated loads can be modified or deleted at this stage based on your judgement.
4. Loads can also be added. With wall A-1 still selected, click on the ***Add…*** button.
5. Select a seismic uniform line load with a magnitude of 10 plf applied to the selected walls only. Click ***OK***.

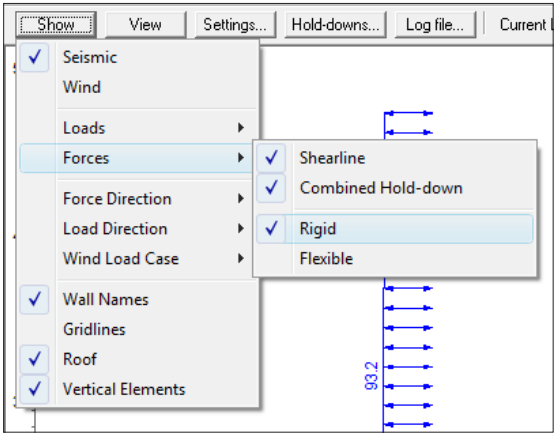


Plan View
To view the various loads and forces applied, use the ***Show*** menu.



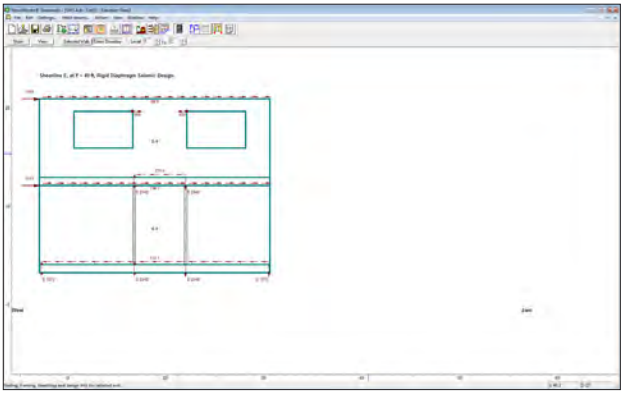
1. Select ***Show/Loads/Masses*** to not show building masses

\



2. Click on the ***Design*** button then return to the ***Loads*** view.

3. Select ***Show/Forces/Rigid*** to view the rigid analysis force distribution.

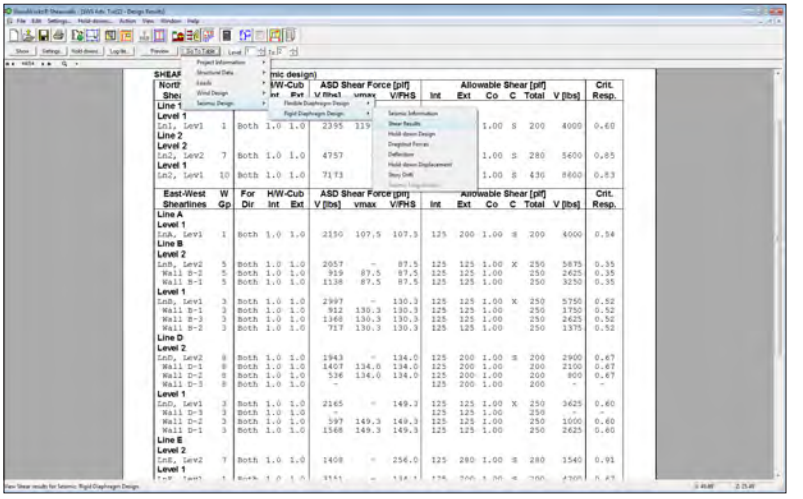


Elevation View

1. Select wall E-1 and click on the Elevation button.
2. Set the levels from 1 to 2 to show both levels at once.
3. Shear forces, hold-down forces and dragstrut forces are shown.
4. Select ***Show/Forces/Hold-downs/Separate*** to view the hold-down force broken-up by its components.

Results

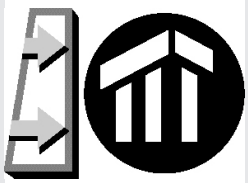
1. Click on the ***Design*** button. The ***Design Results*** view will appear.
2. Scroll up and down to view the results.



3. The ***Show*** button options allow the ***Design Results*** to be customized.
4. Print the results by clicking on the ***Print*** button.

Shearwall Settings

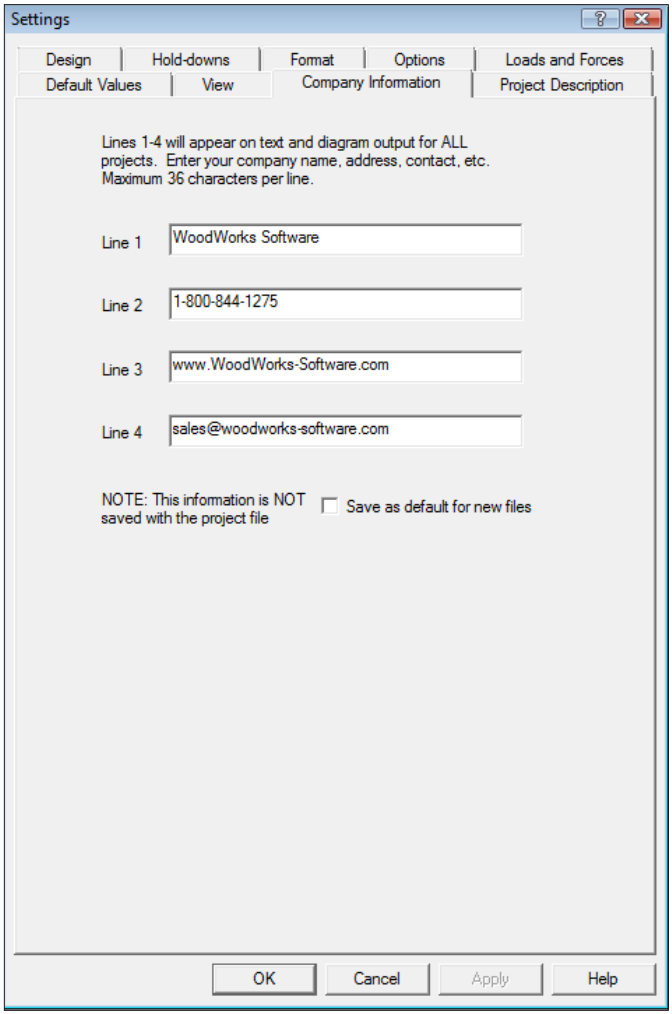
B.4



4.1 General

Click on ***Settings…*** in the toolbar to access the Settings dialog box having tabs for setting data relating to *Design, Format, Options, Loads and Forces, View, Company Information and Project Description.*

Shearwalls provides original settings, however the user should verify the suitability of these default settings and change them as required. For each tab of the dialog box there is a button to *Reset Original Settings*, and a check-box to *Save As Default for New Files* any settings changed by the user.



A general description of these settings is given in the following. Note that the settings for each tab of the dialog box are independent of those for all other tabs.

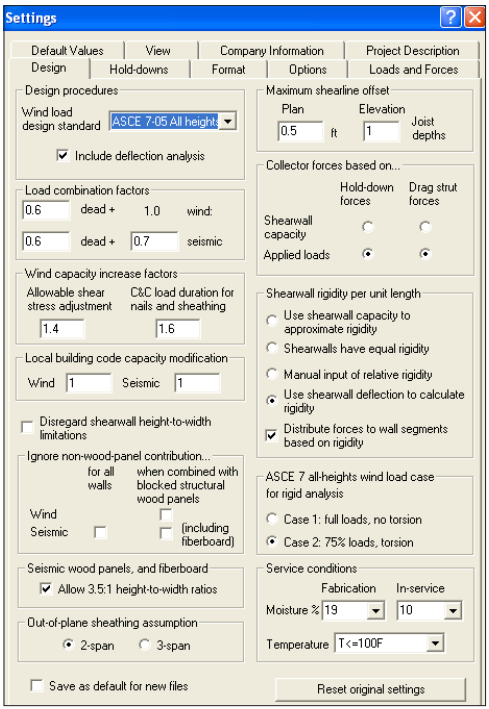
Check out the online tutorial entitled "**Design Settings**" on woodworks-software.com (*Support and Training* Section)

4.2 Design Settings

Design Procedures: Wind Load Design Standard
Use this drop-down to choose which design procedure to follow in the generation of wind loads.

Design Procedures: "Include deflection analysis":
Selecting a deflection analysis not only activates the calculation of shearwall deflections, it also activates the option of distributing forces based on stiffness derived from deflections.

Load Combination Factors (U.S. Only)
Use these boxes to set Allowable Stress Design (ASD) load combination factors for combinations of dead load with seismic or wind loads, as per your building code. Factors may be applied to dead loads to reduce their contribution to overturning resistance in the calculation of hold-down forces. Seismic loads are also normally factored in ASD design to account for the fact that the calculation of E is based on strength principles (i.e. the factor is to convert E from LRFD to ASD).



Wind Capacity Increase Factors (U.S. only)
Various design codes recognize higher resistance values for wood panels for wind design than for seismic design. Use the allowable shear stress adjustment to increase the base shear capacity for MWFRS wind design (The default value of 1.4 conforms to the 40% increase in capacity both IBC and SDPWS allow for wood sheathings). Use the C&C load duration setting to adjust the load duration factor applied in the design of nails and panels subject to C&C suction wind loads. The default value of 1.6 conforms to the NDS load duration factor associated with wind loads.

Local Building Code Capacity modification (U.S. only)
Some local building codes recognize lower allowable shear values for wood panel shearwalls than those listed in model building codes. If required by your building code, use these settings to adjust the allowable shear. This setting may also be used to simulate the overstrength factor:

The software automatically calculates rho and applies it to the load, as per ASCE7-05 12.4.2. In certain circumstances, such as irregular buildings (weak stories 12.3.3.2), users are required to multiply the load by omega instead of the redundancy factor rho. WoodWorks does not include an option of multiplying by omega, but does have a work around solution by reducing the capacity rather than increasing the load. 12.4.3.3 permits the capacity to be increased by 1.2 when the overstrength factor omega is used, and this too can be incorporated in WoodWorks based on the following capacity vs. load equation:

[Capacity x 1.2] = [Load x omega / rho]
This shows we are multiplying the capacity by 1.2, and dividing rho out of the load that WoodWorks puts in, and multiplying omega into the load. But since WoodWorks can only handle the modifications to the capacity, we will reduce the capacity (rho / omega) by the same amount we would have increased the load (omega / rho).

Example:
In the settings/design menu “local building code capacity modification”, seismic cell compute the following, based on omega = 3.0, rho = 1.3, and allowed capacity increase of 1.2:
1.2*(1.3/3.0) = 0.52 (see below)

Note of caution: This method is meant to give a relative comparison of the load vs. capacity. The ratio of load vs. capacity is correct based on this workaround method, however, the value of the load and capacity are both understated by the same amount.

Disregard shearwall height-to-width ratio limitations (U.S. only)
Narrow wood and fibreboard wall segments are generally not considered to contribute to lateral load resistance. By default, Shearwalls implements the height-to-width ratio limitations listed in model building codes for various wall materials subject to wind or seismic loads. Use this setting to disregard these limitations.

Seismic wood panels and fiberboard (U.S. only)
If "*Allow 3.5:1 height-to-width ratios*" is selected, the program considers full height segments with a height-to-width ratio between 2 and 3.5 as shear resisting elements for seismic design. The allowable shear values are reduced as per the NDS.

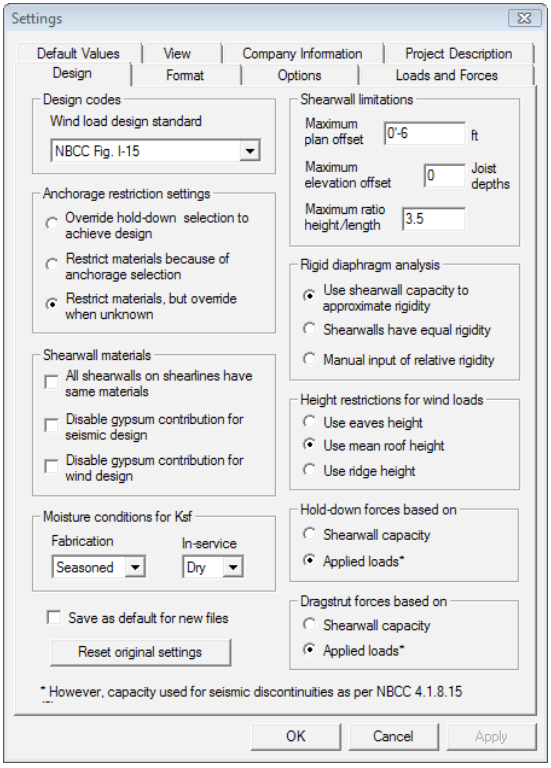
Ignore non-wood panel contribution...
If "*when combined with structural wood panels*" is selected for wind and/or seismic design, then the resistance of two-sided shearwalls will be based only on the allowable shear of wood and fibreboard panels (gypsum and other material will be ignored). This allows higher aspect ratios to be used for wind and / or seismic design for panels where wood or fiberboard is combined with other sheathing materials.

For seismic design, it may be beneficial to ***Ignore non-wood panel contribution*** *for all walls*. This removes the penalty of the increased

seismic loading that would result from the lower response modification factor, R, which must be used when non-wood materials are part of the seismic resistance system.

Out of plane sheathing assumption: The Special Design Provisions for Wind and Seismic (SDPWS) out-of-plane sheathing values are based on a 2-span configuration. Other publications, such as the Wood Frame Construction Manual, lists these values based on sheathing supported over 3 spans. Furthermore note under SDPWS table 3.2.2 allows for an increase of values if the sheathing is continuous over 3-spans. Selecting the 3-span here increases the tabulated values by the intended 25%.

Service Conditions
Use this setting to specify temperature and moisture conditions. Shearwalls will apply the appropriate design factors.



Anchorage Restriction Settings (Canada only)
Use the radio buttons to choose one of the following options:

i. **Override hold-down selection to achieve design**
The program allows a choice of all materials in the design of shearwalls. It specifies hold-downs if the required shear capacity of a wall cannot be achieved without them, using the wall materials selected by the user. If wall materials are selected that always require hold-downs, the hold-

down setting automatically changes to *Hold-downs all segments*.

ii. Restrict materials because of anchorage selection
The program restricts the materials available for the design of shearwalls to those that can achieve the required shear resistance with the hold-down configuration specified by the user. If the user leaves a wall material selection *unknown*, the program still limits the wall design in accordance with the material restrictions.

iii. Restrict materials, but override when unknown
The program restricts materials the same way as in option ii, with the following exception: if the user leaves a wall material selection as *unknown*, the program will not limit this parameter in its design of shearwalls. It will override the user-specified hold-down configuration to achieve the required shear capacity, rather than limit the *unknown* parameter.

Shearwall Limitations
The *Maximum Plan Offset* defines the distance up to which shearwalls on the same level may be apart and still be considered part of the same shearline.

The *Maximum Elevation Offset* defines the distance up to which shearwalls on different levels may be apart and still be considered part of the same shearline.

Hold-down / Dragstrut Forces based on...
The hold-down and dragstrut forces may be based either on the presumably higher capacity of the associated shearwall or on the calculated value of the corresponding hold-down force, that is, the *applied loads*.

Rigid Diaphragm Analysis (Canada) / Shearwall Rigidity per unit length (U.S.)
There are four methods to establish the relative rigidity of shearwalls for a Rigid Diaphragm Analysis: by using shearwall capacity to approximate rigidity; by considering that all shearwalls have equal rigidity; by manual input of relative rigidity; or using deflection derived stiffness.

It is possible to select either the shearwall capacity or deflection option, then adjust rigidities using the manual method. This method can help simulate proprietary shearwall sections. The method of distributing loads within the shearline will be the same as the method chosen to distribute loads to the shearlines

Distributed Forces to Wall Segments Based on Rigidity (U.S. Only)
Selecting this option distributes the shear load within the shearline based on individual wall rigidities, rather than distributing the shear uniformly along all walls within the shearline. This option does not effect the total shear distributed to the shearline.

ASCE 7 all heights wind load case for rigid analysis (U.S. only)

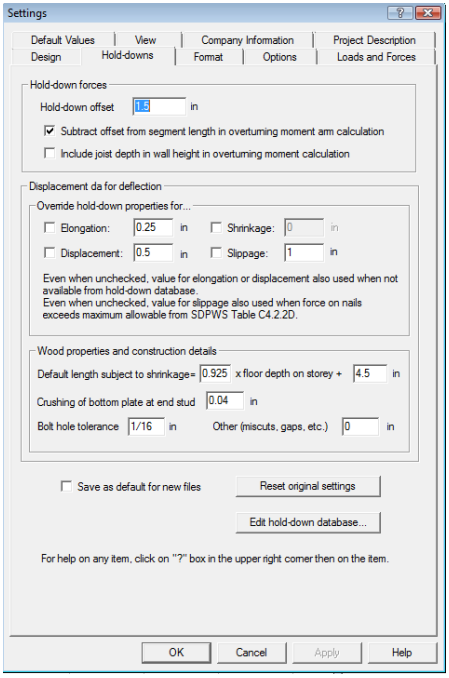
Case 1: full wind loads without accounting for torsional effects will be used if this option is selected for rigid analysis.

Case 2: 75% of wind loads plus the effect of torsion will be used for rigid analysis.

Both options can potentially govern, depending on the shape of your building, and therefore the design results for both cases should be reviewed.

4.3 Hold-Down Settings

The *Hold-down offset* will be used in the calculation of hold-down forces if *Subtract offset from segment length...*" is selected. Left unchecked, the moment arm will be based on the total distance from end to end of the segment, resulting in lower forces in tension and compression at the ends of the panel .



In the US, the joist depth is not included in the overturning calculation, as shown in the SDPWS. However, some engineers prefer to take the more conservative approach of including this extra height, and selecting "*Include joist depth in wall height...*" will give this extra flexibility. The Canadian version automatically includes the floor depth as per the CSA O86.

Displacement da for deflection:
If the boxes are checked in the *Displacement da for deflection* section, the user can input values for the elongation, displacement, shrinkage and slippage that will override the hold-down properties given in the hold-down database. These values will be used for all the hold-down in the structure.

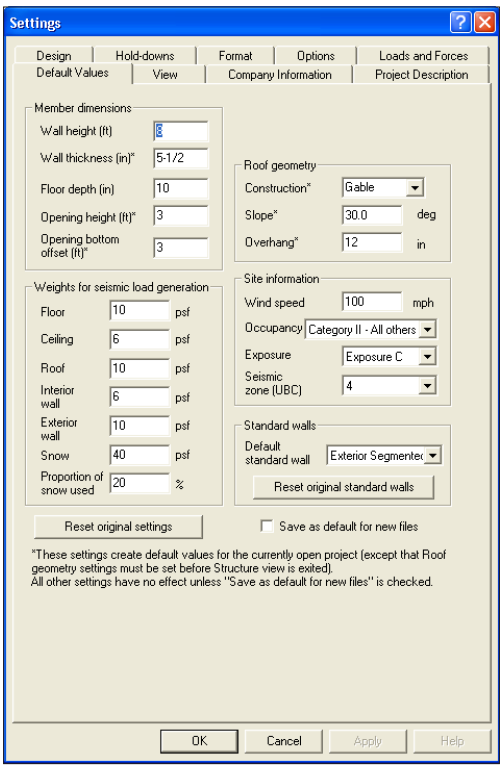
The *Wood properties and construction details* section indicates the different sources for vertical shearwall displacement at the hold-downs due to wood shrinkage, crushing of wood, slippage in bolt hole and construction quality and allows the user to input specific values.

From the hold-down settings, you can reset the original settings and access the hold-down database This database contains pre-existing hold-downs and allows you to create new hold-downs.

4.4 Other Settings

Default Values Settings
Default values that are used with all new files can be specified for member dimensions, self-weights for seismic building mass determination, roof geometry, site information for load generation, and the standard wall type to be used.

Format Settings
It is possible to set the *Unit System*, *Imperial* (English) *Formatting* and the *Font Size* for both *Printer* and *Screen.*



Options Settings
A number of options exist to *Display* information and data, and to set the default standard wall used when creating new walls.

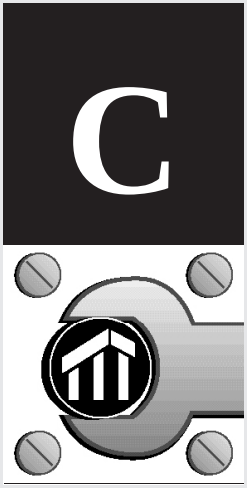
Loads and Forces Settings
It is possible to selectively display *Loads and Forces* in *Plan View* and *Elevation View*, as well as to selectively display *Design Results*.

View Settings
It is possible to set the *Limits* of the *Viewing Area*, the *Snap Increment* (it can only be decreased, and once decreased it can not be increased), and the intervals that gridlines are displayed, if at all.

Company and Project Settings
Company Information can be displayed on text and diagram output for all projects. A specific *Project Description* can be displayed for individual projects.

The same settings can be set with the data bar show button. This tab should be used to overhaul the configuration of loads and forces you wish to view.

Connections



1 What is Connections?

About Connections

The WoodWorks® Connections software will be referred to as Connections throughout this section.

This is meant to give a brief introduction to the features and on how to use Connections. This brief introduction to Connections should be sufficient to help you become an expert user of the software. However, technical support is available if you have further questions.

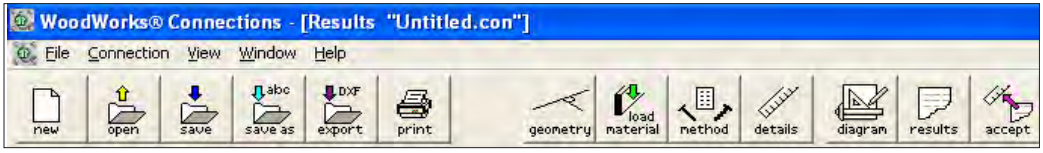
Connections is made-up of several different screens which are followed in sequence to complete the design of a connection. You proceed through these screens using Drop-Down Menus and toolbars.

Drop-Down Menus

The top of the Connections window always contains a drop-down menu bar.
To make a menu selection:

- move the mouse to point to a menu bar item
- press the left mouse button down
- while holding the mouse button down, move to the appropriate drop-down menu item
- release the mouse button

An alternative selection method is to click the mouse (press and release the left mouse button rapidly) on a menu bar item. This causes the drop-down list to drop down and stay. Now click on the appropriate sub-menu item to make your selection.



Toolbars

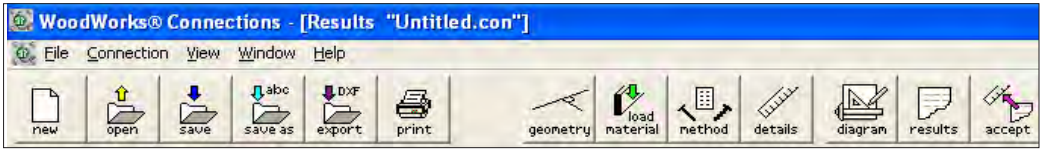
Connections includes toolbars with toolbar buttons for the main procedures of the program. When performing a design from start to finish, you should proceed from left to right along the toolbar. The toolbar buttons are placed in a sequential order to facilitate designing.

Almost all of Connections features can be accessed through toolbars. This speeds-up the design process by minimizing the use of drop-down menus.

toolbars are placed near the top of the Connections window directly under the drop-down menu bar.

To make a selection using a toolbar:

- move the mouse to point to a toolbar button
- press the left mouse button down
- release the mouse button



Toolbar Buttons

The following describes the main features of each of the toolbar buttons in Connections.

New
Click on the ***New*** button in the toolbar to a start a new connection design file.

Open
Click on the ***Open*** button in the toolbar to retrieve an existing file. This command can be used to edit an existing file or to use an existing file as the basis for a new design.

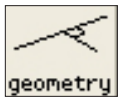
Save
Click on the ***Save*** button in the toolbar to save the current design.

Save As
Click on the ***Save As*** button in the toolbar to save the current design but with a different name.

DXF Export
Selected fasteners can be exported for use in CAD.

Print
Click on the ***Print*** button in the toolbar to print the diagram or the design results. These screens must first be open.

Geometry
Click on the ***Geometry*** button in the toolbar to change to the ***Geometry*** view. In this view, you can change the current connection geometry.



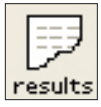
Load Material
Click on the ***Load Material*** button in the toolbar to change to the ***Load Material*** view. In this view, you can specify the applied load and the material parameters.

Method
Click on the ***Method*** button in the toolbar to change to the ***Method*** view. In this view, you can change specify the type of connector to be used.

Details
Click on the ***Details*** button in the toolbar to change to the ***Details*** view. In this view, you can specify connector parameters.

Diagram
Click on the ***Diagram*** button in the toolbar to change to the ***Diagram*** view. This view displays the current connection detail diagram.

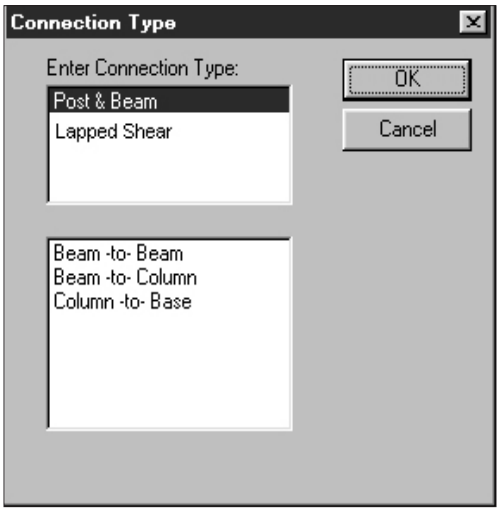
Results
Click on the ***Results*** button in the toolbar to change to the ***Results*** view. This view displays the design results output.



Accept
Click on the ***Accept*** button to accept the current design results and to input this information into the details screen.

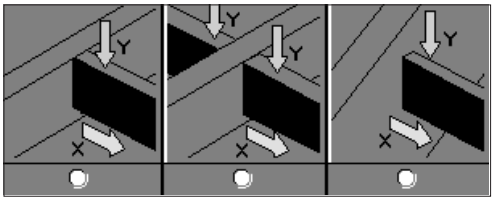
Connection Types

Connections can design for both ***Post & Beam*** and ***Lapped Shear*** connection types.

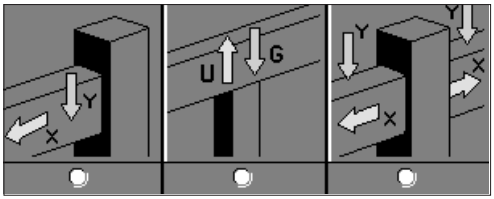


Post & Beam Connections
Post & Beam connections include the following configurations:

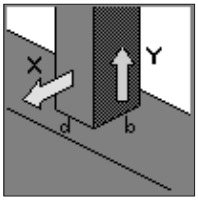
- Beam-to-Beam



- Beam-to-Column

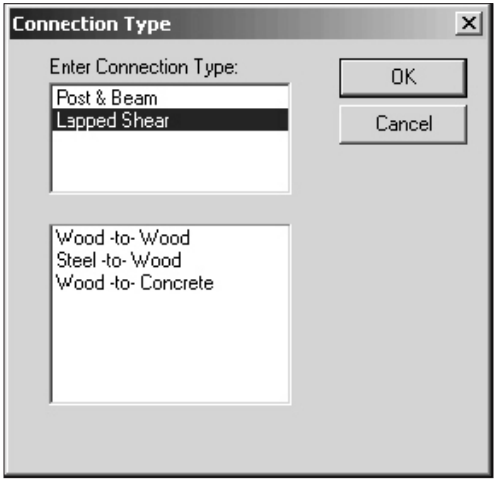


- Column-to-Base



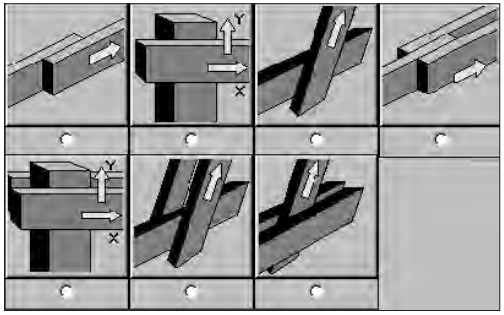
The ***Post & Beam*** connections can be designed using any of the following connector types:

- Bolts
- Shear Plates
- Rivets (Canadian only)
- Heavy-duty Hangers
- Lag screws

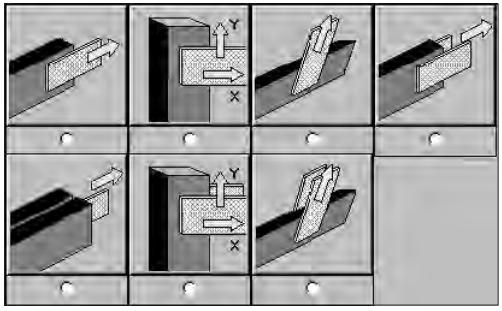


Lapped Shear Connections
Connections designs for the following Lapped Shear connections using nails or bolts.

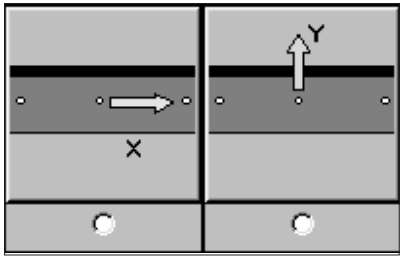
- Wood-to-Wood



- Wood-to-Steel



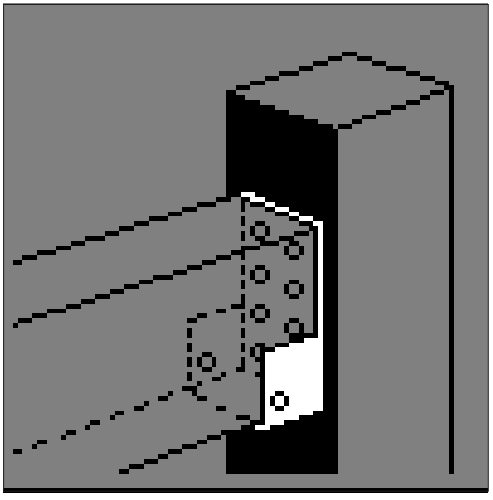
- Wood-to-Concrete



2 Post & Beam Connections

Introduction

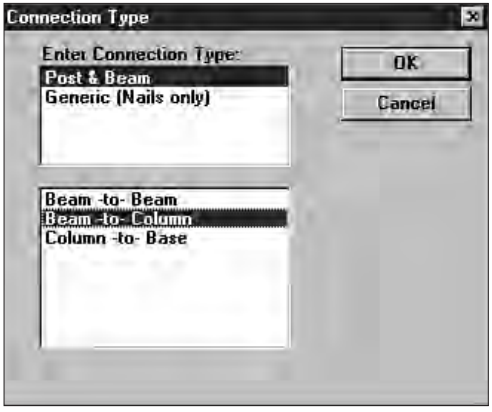
In this tutorial you will create and design a bolted bottom hanger for a post to beam type connection as shown below.



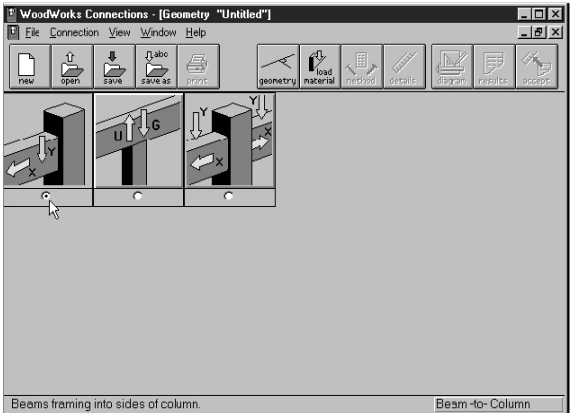
Connection Type

1. Click the ***New*** button on the toolbar.

2. Select the connection type ***Post & Beam*** with a ***Beam-to-Column*** configuration.



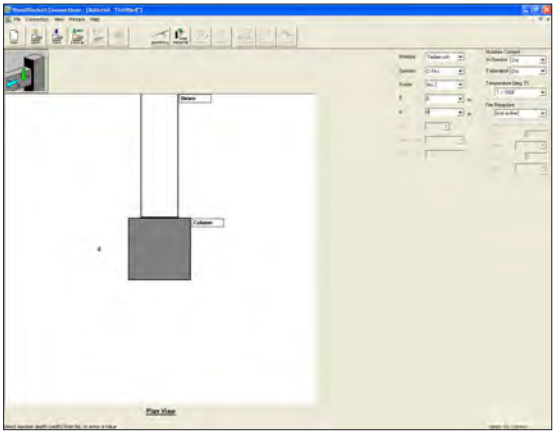
3. Click on ***OK***.



Connection Geometry

1. Click on the first (left most) connection geometry, as shown.

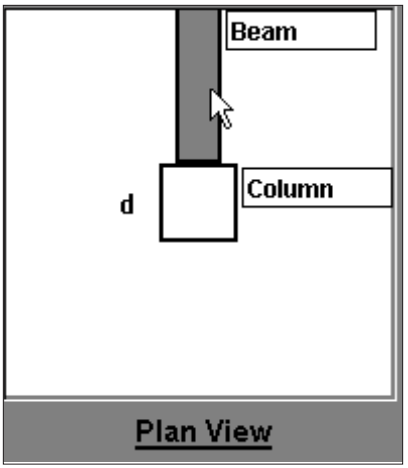
   Note that a description of the connection geometry appears on the status bar.



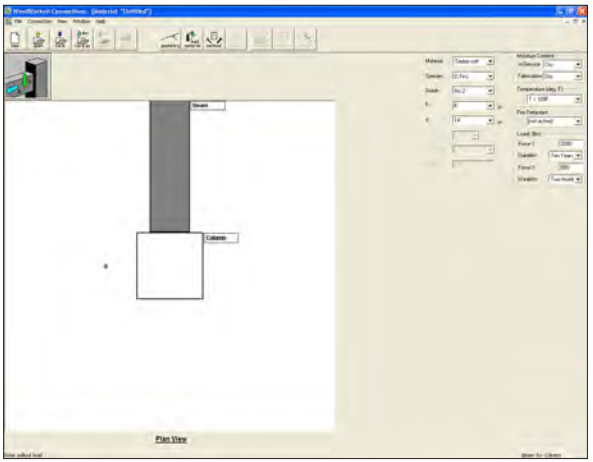
Load and Material Details

1. Click on the ***Load/ Material*** button.

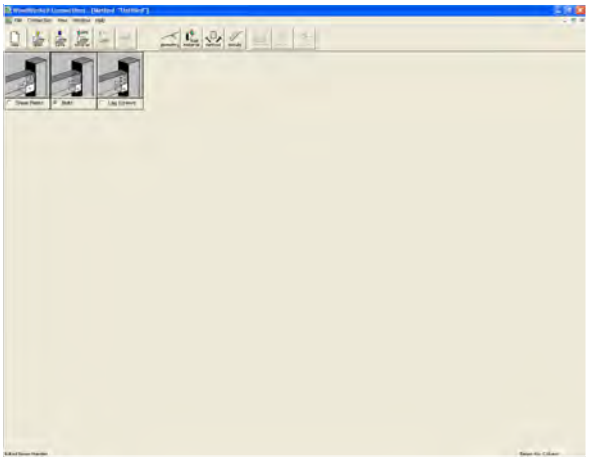
2. Enter a width, b, of 8" and a depth, d, of 8" for the column.



3. Click on the beam which is shown graphically in the plan view. This allows you to specify the beam parameters.



4. Specify a width, b, of **6"** and a depth, d, of **14"** for the beam.
5. Enter a ***Force Y*** of 3000 lbs and a ***Force X*** of 800 lbs.

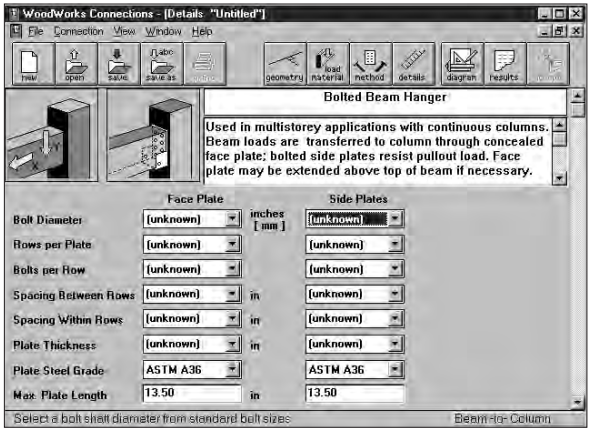


Connector Type

1. Click on the ***Method*** button.

2. Select ***Bolts*** as the connector type by clicking on the ***Bolts*** graphic.

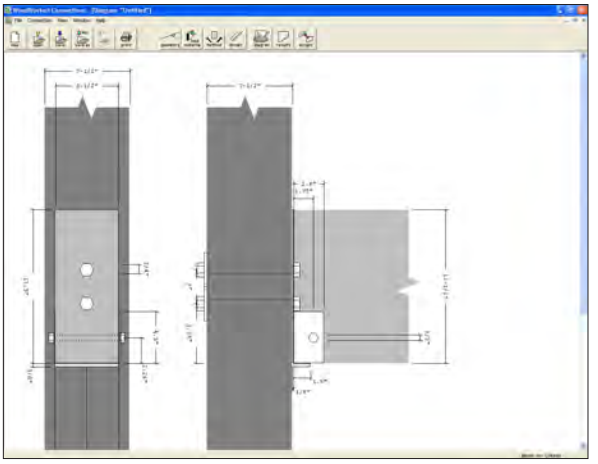
3. For selected connectors exporting dxf files of the connection drawing for use in AutoCAD can be done. Three separate dxf files will be exported, approriate for:
   - shop drawings
   - assembly drawings
   - combined, exporting all layers



Connector Details

1. Click on the ***Details*** button.

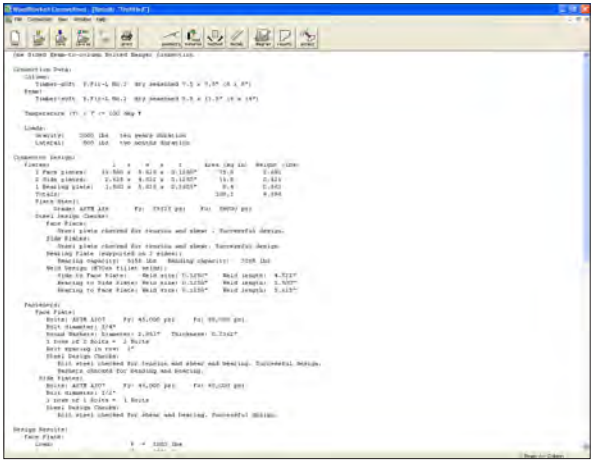
At this point you can specify details about the connection such as bolt diameter, plate thickness, or spacing. You can also leave some or all of the details as ***unknown*** and allow Connections to choose these parameters. In this example, leave everything as ***unknown*** to start.



Connection Detail

1. Click on the ***Diagram*** button.

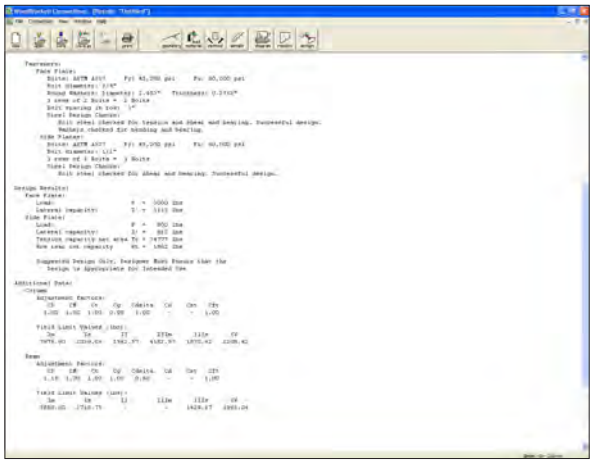
2. To print the connection detail, click on the ***Print*** button.



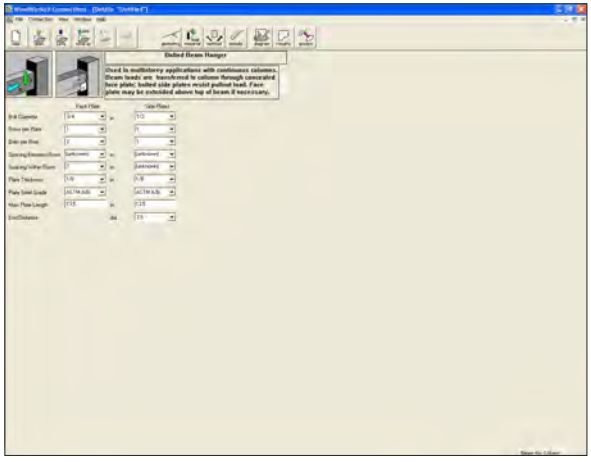
Design Results

1. Click on the ***Results*** button to view the design results for the connection.

2. To print the design results, click on the ***Print*** button.

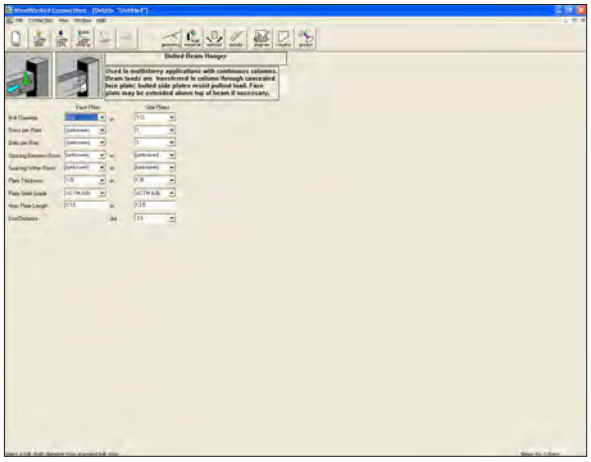


3. You can scroll up and down to view more of the design results output.



4. Click on the ***Accept*** button. This will automatically accept the parameters chosen by Connections during the design and will place you in the ***Details*** view.

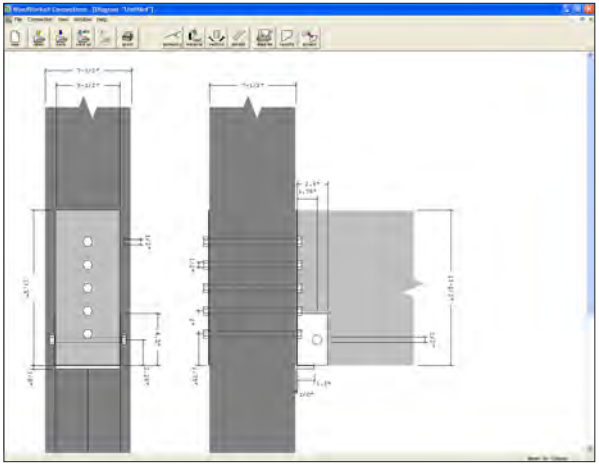
You will notice that all of the parameters now have specified values (***Spacing Between Rows*** and ***Spacing Within Rows*** will remain as ***unknown*** when there is only one row of connectors or only one connector required).



Re-Designing the Connection

If at this point you are not satisfied with the suggested design, you can change some parameters in the ***Details*** view and re-design the connection.

1. Change the ***Bolt Diameter*** to 1/2" for the Face Plate.
2. Change all of the remaining Face Plate parameters to ***unknown*** except for the plate thickness. Leave this as 1/8".



3. Click on the ***Diagram*** button.

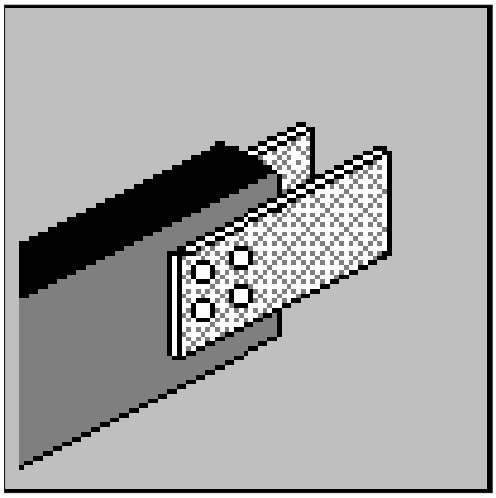
4. Click on the ***Results*** button.

5. Repeat the above steps while changing load conditions, connection geometry, connector details or any other parameters. This will allow you to quickly become familiar with Connections.

3 Lapped Shear Connections

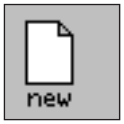
Introduction

In this tutorial you will create and design a bolted wood to steel splice connection as shown below.

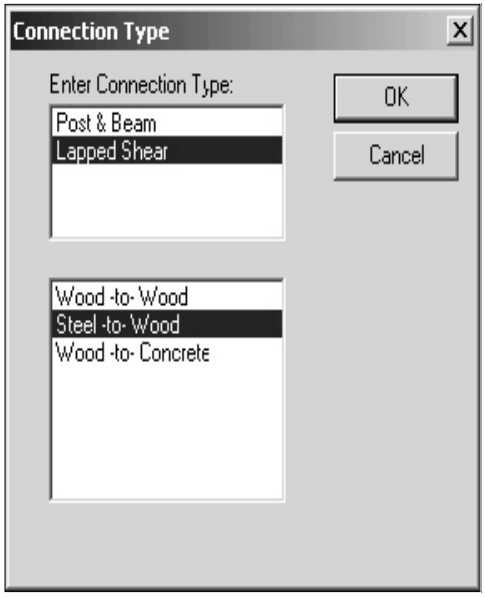


Connection Type

1. Click the ***New*** button on the toolbar.

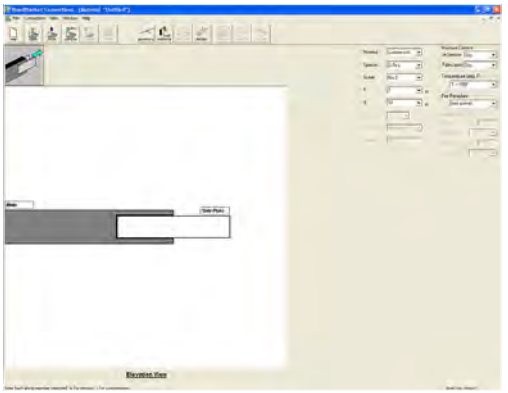


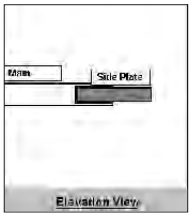
2. Select the connection type ***Lapped Shear*** with a ***Wood-to-Steel*** configuration.

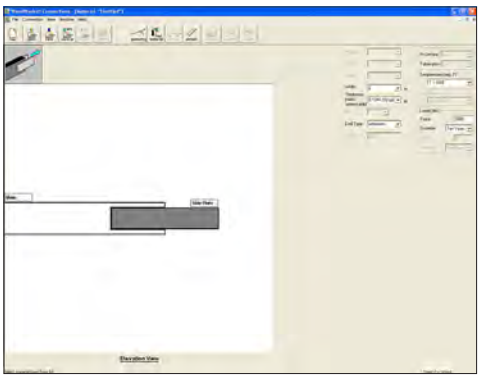


Connection Geometry

1. Click on the geometry which represents a double shear splice connection (fourth image), as shown.







Load and Material Details

1. Click on the ***Load/Material*** button.

2. Enter a width, b, of 2 in. And a depth, d, of 10 in. for the main wood member.
3. Click on the steel side plate which is shown graphically in the elevation view. This allows you to specify the width of the steel side plate.
4. Specify a width of 6 in. (Note that the thickness of the plate for bolted connections is specified in the details screen.)
5. Enter a force of 3000 lbs.

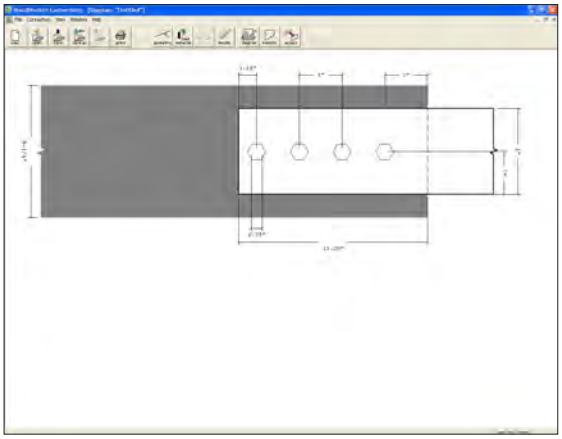
Connector Type

1. Note that the connector type button is grayed out. Only bolted connections are available for this configuration. Proceed to connector details.

Connector Details

1. Click on the details button.

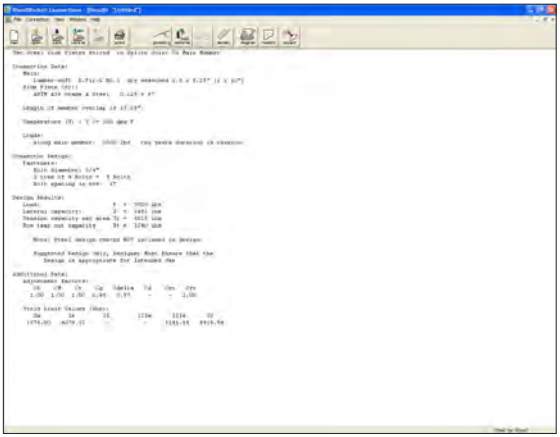
At this point you can specify details about the connection. You can also leave some or all of the details as ***unknown*** and allow Connections to choose these parameters. In this example, leave everything as ***unknown*** to start.



Connection Detail

1. Click on the ***Diagram*** button.

2. To print the connection detail, click on the ***Print*** button.

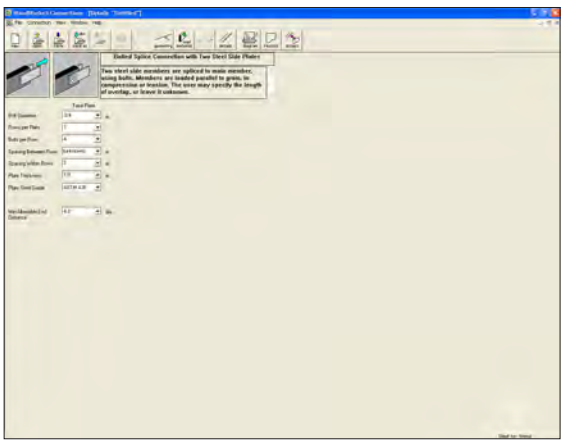


Design Results

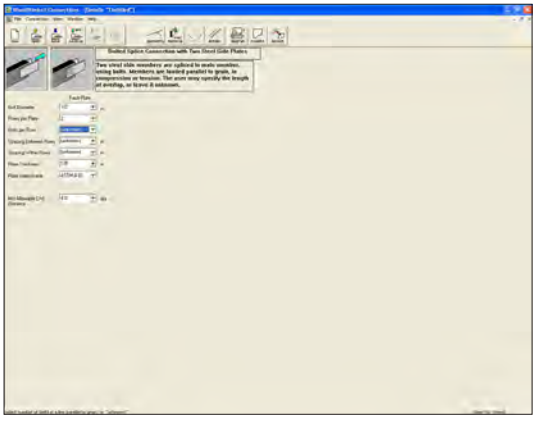
1. Click on the ***Results*** button to view the design results for the connection.

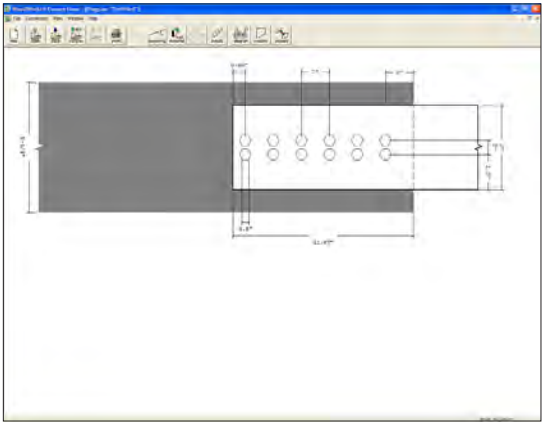
2. To print the design results click on the ***Print*** button.

3. You can scroll up and down to view more of the ***Results*** output.



4. Click on the ***Accept*** button. This will automatically accept the parameters chosen by Connections during the design and place you in the ***Details*** view.





Re-Designing the Connection

If at this point you are not satisfied with the suggested design, you can change some parameters in the ***Details*** view and re-design the connection.

1. Change the ***Bolt Diameter*** to 1/2 inch**.**
2. Change the ***Rows per Plate*** to 2.
3. Change the ***Bolts per Row*** to ***unknown.***
4. Click on the ***Diagram*** button.

The Database Editor

1 What is the Database Editor?

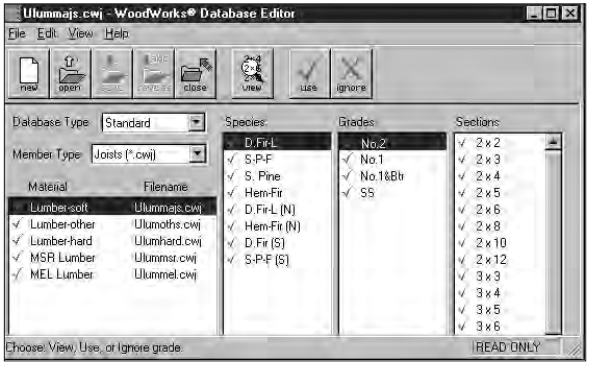
The Database Editor is a simple application that allows you to create new material databases, view existing databases, and customize the material database list for local availability.

Scope of Database Editor

The Database Editor is currently limited to operating on databases of wood FRAMING materials only. This includes beams, columns, joists and wall studs consisting of solid sawn, glulam and engineered wood products. Sheathing materials and fasteners have not been incorporated at this time.

Databases used by the Design Office programs:

- **Sizer** — all databases
- **Connections** — all databases except I-joists, LVL, PSL materials
- **Shearwalls** — only the wall stud databases



Note:
1. ALL the Design Office programs use a common set of databases that can be viewed and modified by the Database Editor.
2. Sizer exploits all features of the Database Editor.
3. Connections and Shearwalls do not skip Species, Grades and Sections that have been set to "Ignore" by the Database Editor.
4. A change to any database will affect all programs that use it (with the exception of item 3).

File Buttons

New

Click on the ***new*** button in the toolbar to a create custom database file. The program will prompt you with a series of dialog boxes to walk you through the process of creating a custom database file.

Open

Click on the ***open*** button in the toolbar or click on the database file name in the file list to retrieve an existing database. This command retrieves an existing material database file. You can specify the type of database, such as Beams, Columns, Joists, and Wall Studs.

Save

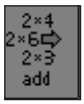
Click on the ***save*** button in the toolbar to update a material database which you applied changes to. The ***save*** button also updates the current ***use*** and ***ignore*** settings for customizing the database list.

Save As
Click on the ***save as*** button in the toolbar to make a copy of the current database but with a different name.

Close
Click on the ***close*** button in the toolbar to close the current database without saving any changes which may have been done to this file.

Editing and Viewing Buttons

Add
Click on the ***add*** button in the toolbar to add new species, grades or sections to a Custom database file. The ***add*** button is only available for Custom database files. A maximum of three species, three grades, and six sections is allowed for Custom database files.



Delete
Click on the ***delete*** button in the toolbar to delete existing species, grades or sections in a Custom database file. The ***delete*** button is only available for Custom database files.

Edit
Click on the ***edit*** button in the toolbar to edit the species, grade or section properties of a Custom database file. The ***edit*** button is only available for Custom database files.

View
Click on the view button in the toolbar to view the species, grade or section properties of a Standard database file. The view button is only available for the Standard database files.

Customizing Buttons

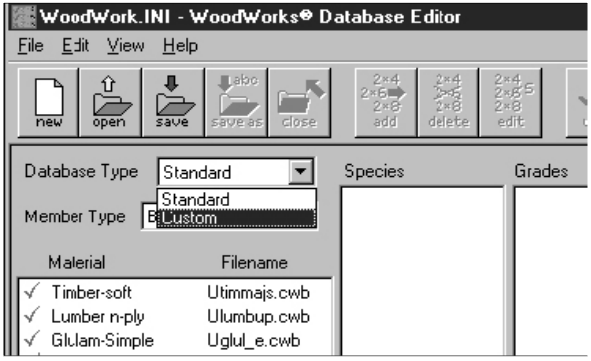
The customizing buttons allow you to select those material databases that will be used or ignored during the design process. **This type of customization affects Size**r **and Shearwalls only**. Connections disregards the ignore settings and can use all of the material databases.

Use
Click on the ***use*** button in the toolbar to allow a selected species, grade, or section of a material database to be used by Sizer and Shearwalls during the design process. A check mark will appear beside the species, grade or section to indicate that it can be used by Sizer and Shearwalls. The ***use*** button applies to both Standard and Custom databases.

Ignore
Click on the ***ignore*** button in the toolbar to prevent Sizer and Shearwalls from using a selected species, grade or section during the design process. An 'X' will appear beside the species, grade or section to indicate that it can not be used by Sizer and Shearwalls. The ***ignore*** button applies to both Standard and Custom databases.

2 Material Databases

Material databases are described as either being a Standard database or a Custom database. The properties of the Standard databases can be viewed, but not edited. Custom database can be viewed, edited and created.



The ***Database Type*** drop-down list allows you to switch between the Standard and Custom databases.

Standard Material Databases

Standard material databases include sawn timber and glulam members whose strength properties are based on those published in the applicable design standards. The Standard databases are based on values published in the National Design Specification (NDS) for Wood Construction for the U.S. and based on the CSA O86.1 for Canada.

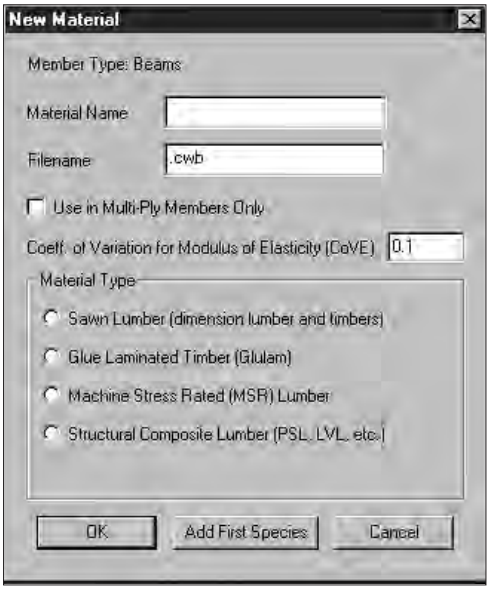
Custom Material Databases

Custom material databases include I-joists, Parallel Strand Lumber (PSL), Laminated Veneer Lumber (LVL), and Laminated Strand Lumber (LSL) as a default. Generic values for the strength properties have been included in these databases. Since these are generic values, verification of these properties should be done to ensure that they are applicable to the material you wish to design.

Warning: Custom databases included with WoodWorks or created by the user should only be used for preliminary sizing of members. Contact the engineered wood product manufacturer directly for an accurate and complete design of proprietary wood products.

Warning: You can retain database customizations made with WoodWorks® Database Editor from an existing installation by specifying the same installation folder as the existing installation, choosing the ***Custom Install*** *Setup Type*, then unchecking ***Custom Materials Database*** in *Select Components*. Failing to remove the checkmark will result in your custom database being overwritten by the new custom database.

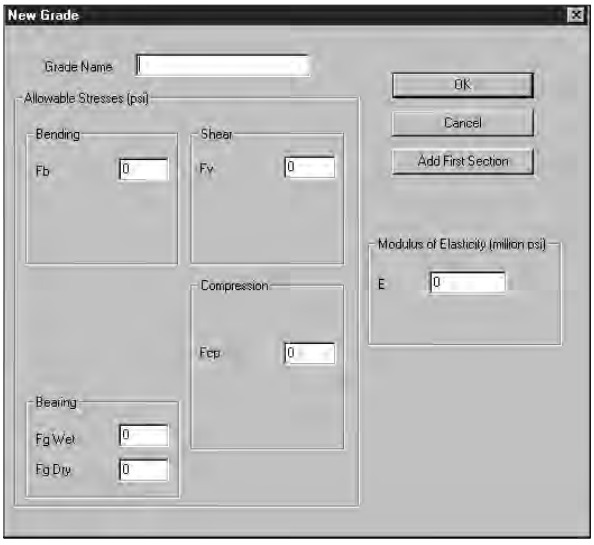
3 Creating Custom Databases



The Database Editor allows you to create custom material databases for beams, joist, columns, and walls. You can specify material type, species, grade, dimension, and strength properties.

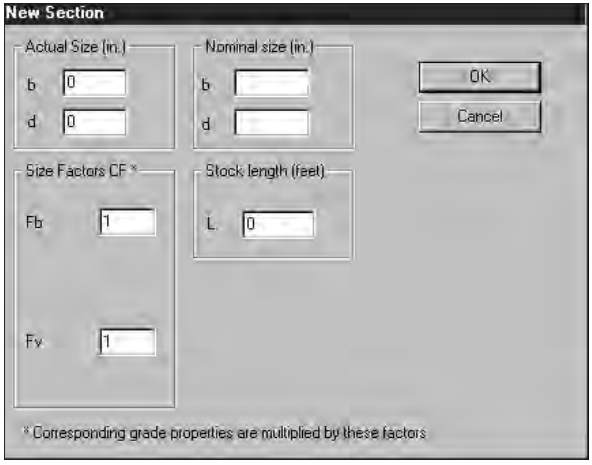
Material
Click the ***new*** button on the toolbar menu. The ***New Material*** dialog opens and prompts you to enter a database name, specify the type of material, define the COV_E (if applicable) and define whether this is a multi-ply member or not.

Species
Click on ***Add First Species*** in the New Material dialog. Now the New Species dialog automatically opens and prompts you to enter a species name, the specific gravity of the material (for self-weight calculations), and a southern pine option for glulam.



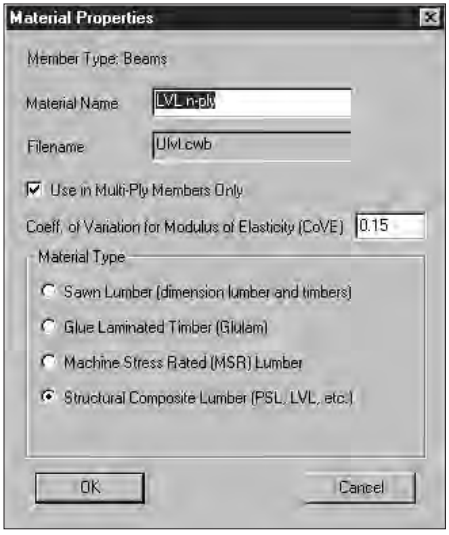
Grade

Click on ***Add First Grade*** in the New Species dialog. Now the ***New Grade*** dialog automatically opens and prompts you to enter a grade name, the allowable stress values, the modulus of elasticity (E), and to specify the material use as a beam, post or both.



Section

Click on ***Add First Section*** in the New Grade dialog. Now the New Section dialog automatically opens and prompts you to enter the actual and nominal dimensions, the size factors, the stock length, and the material shape as either beam, post or both.

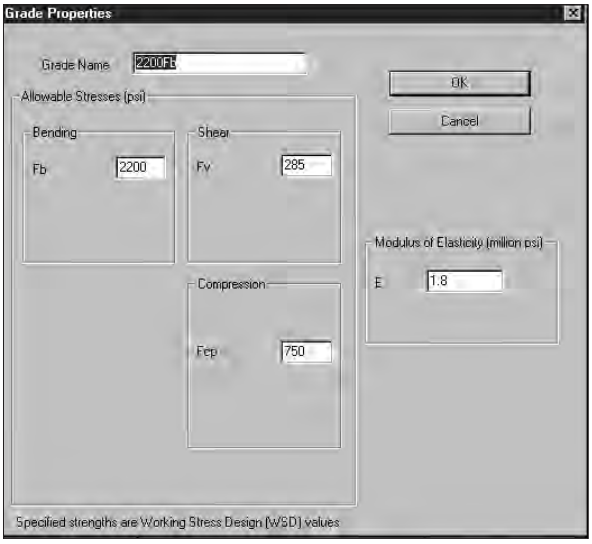


Modifying Custom Databases

The Database Editor allows you to modify Custom database files by adding, deleting or editing the material, species, grade or section properties.

The following steps describe how to modify a custom database file.

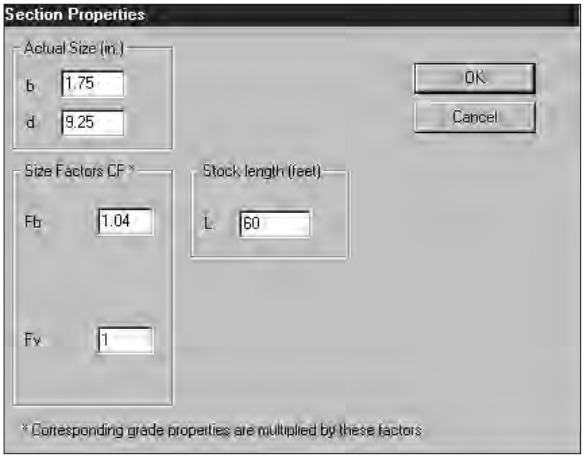
Step 1
You must first open the Custom database file that you want to modify. This is done by using the ***open*** button or by clicking on the file name in the file list.



Step 2
You must now select the property that you want to edit. This is done by highlighting the material, species, grade or section that you wish to edit and then clicking on the ***add***, ***delete***, or ***edit*** button from the toolbar.

Depending on the information that you are editing, one of the four following dialog boxes will open:

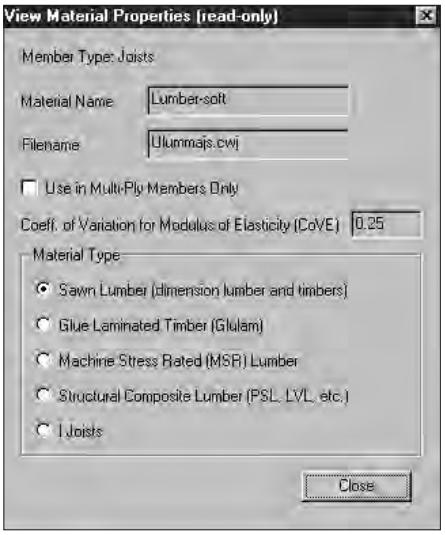
- *Material Properties* dialog
- *Species Properties* dialog
- *Grade Properties* dialog
- *Section Properties* dialog



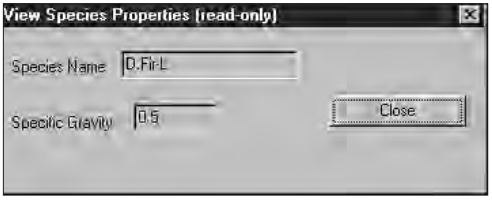
Step 3
Click on the ***save*** button to save any of the changes.

> Note: Installing updates of the custom database will erase any modifications made previously. Remove checkmark from custom database during the installation process in order to retain your original copy of the custom database.

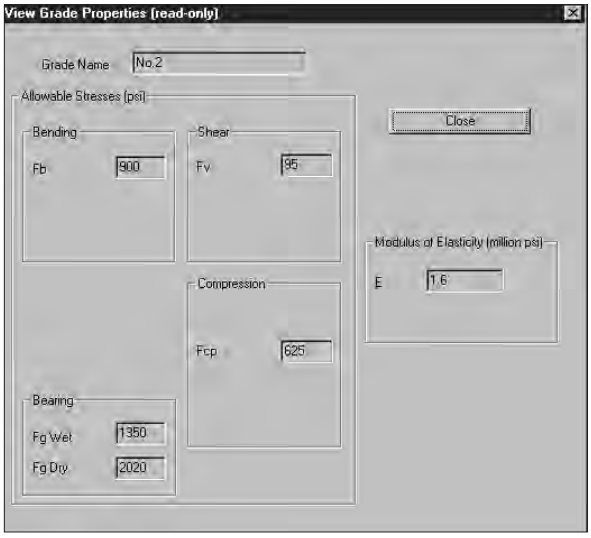
4 Viewing Standard Databases

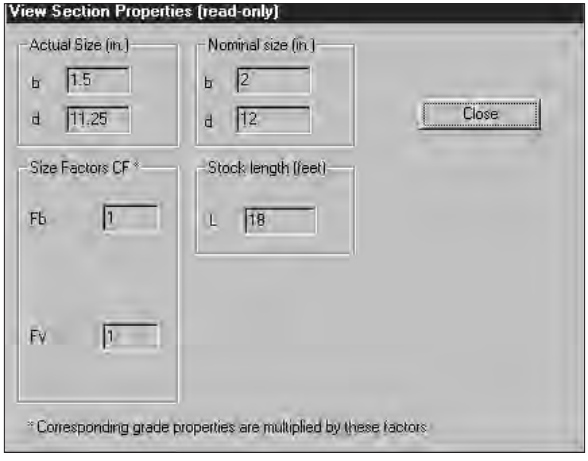


The Database Editor allows you to view the material, species, grade and section properties of the Standard databases. The following steps describe how to do this.



Step 1
You must first open the Standard database file that you want to modify. This is done by using the ***open*** button or by clicking on the file name in the file list.





Step 2
You must now select the property that you want to view. This is done by highlighting the material, species, grade or section that you wish to view and then clicking on the *view* button from the toolbar.

Depending on the information that you are viewing, one of the four following dialog boxes will open:

- Material Properties dialog
- Species Properties dialog
- Grade Properties dialog
- Section Properties dialog

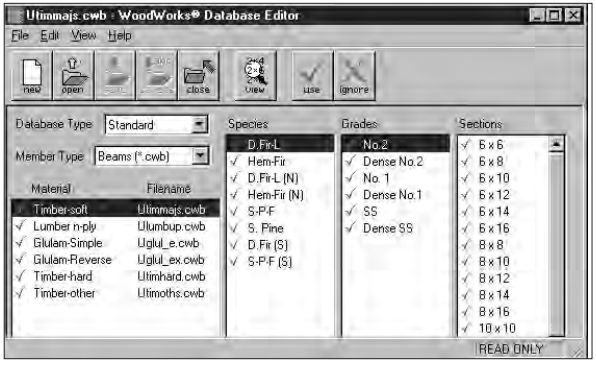
Step 3
Click on the *save* button to save any of the changes.

5 Customizing the Database List (Sizer Only)

The database editor includes a feature which will allow you to customize the entire database list for your needs by designing for only certain materials, species, grades, or section sizes.

This type of customization only affects Sizer. Connections and Shearwalls disregard the ***ignore*** settings and can use all of the material databases.

The customizing feature simply places a '✓' or an '✗' beside the material, species, grade, or section to let Sizer know which properties to ***use*** or ***ignore*** during the design process. These properties are not erased by the customizing feature and can therefore be retrieved at a later time. Sizer initially sets all properties to ***use*** (✓) as a default.



Customizing

The following steps describe how to customize your database list.

Step 1
You must first open the database file (either Standard or Custom) that you want to customize. This is done by using the ***open*** button or by clicking on the file name.

Step 2
Click on the material(s), species, grade(s), or section(s) that you wish to customize so that it is highlighted (Note that multiple selections can be made at once by clicking while holding down the CNTRL or SHIFT key).

Step 3
Click on the ***use*** button (✓) to use this selection during the design process or click on the ***ignore*** button (✗) to ignore this selection during the design process.

Step 4
Click on the ***save*** button to save any customizing done to the materials list.

Important note: Clicking the ***save*** button at this point will also save any changes done to an open Custom database file.